inside
progeCAD
2017

The tutorial for new users by Ralph Grabowski

progeCAD
Your Power, right away

ProgeSOFT

Copyright Information

Table of Contents

2 — Your Guided Tour Through progeCAD 9

3 − 141 Tips 47

10 — Creating Symbols

11 — Placing Text

Welcome to progeCAD

Welcome to progeCAD!

By learning how to work with progeCAD, you are joining the many who use software to draft drawings. progeCAD is exciting software, because it has many of the features of CAD software that could cost ten times as much — yet it is similar to the market leader, AutoCAD.

About progeCAD

progeCAD is based on IntelliCAD, and comes from progeSOFT (http://www.progesoft.com).

One of the main benefits to progeCAD is its GUI (graphical user interface), which is designed to have an "AutoCAD-like" feeling, as shown by the figure on the following pages. To view a list of the similarities and differences between progeCAD and AutoCAD, please check out http://www.progesoft.com/en/products/progecad-professional/#compare.

ogeCAD 2017 Professional - [Drawing1.dwg]

Tools   EasyArch3D   EasyArch Extras   Help

0

BYLAYER
BYLAYER
BYLAYER

Layer

Match
Properties

Distance

Paste

Layers   Properties ▾   Utilities ▾   Clipboard

464.6242,222.9732,0   1:1   MODEL
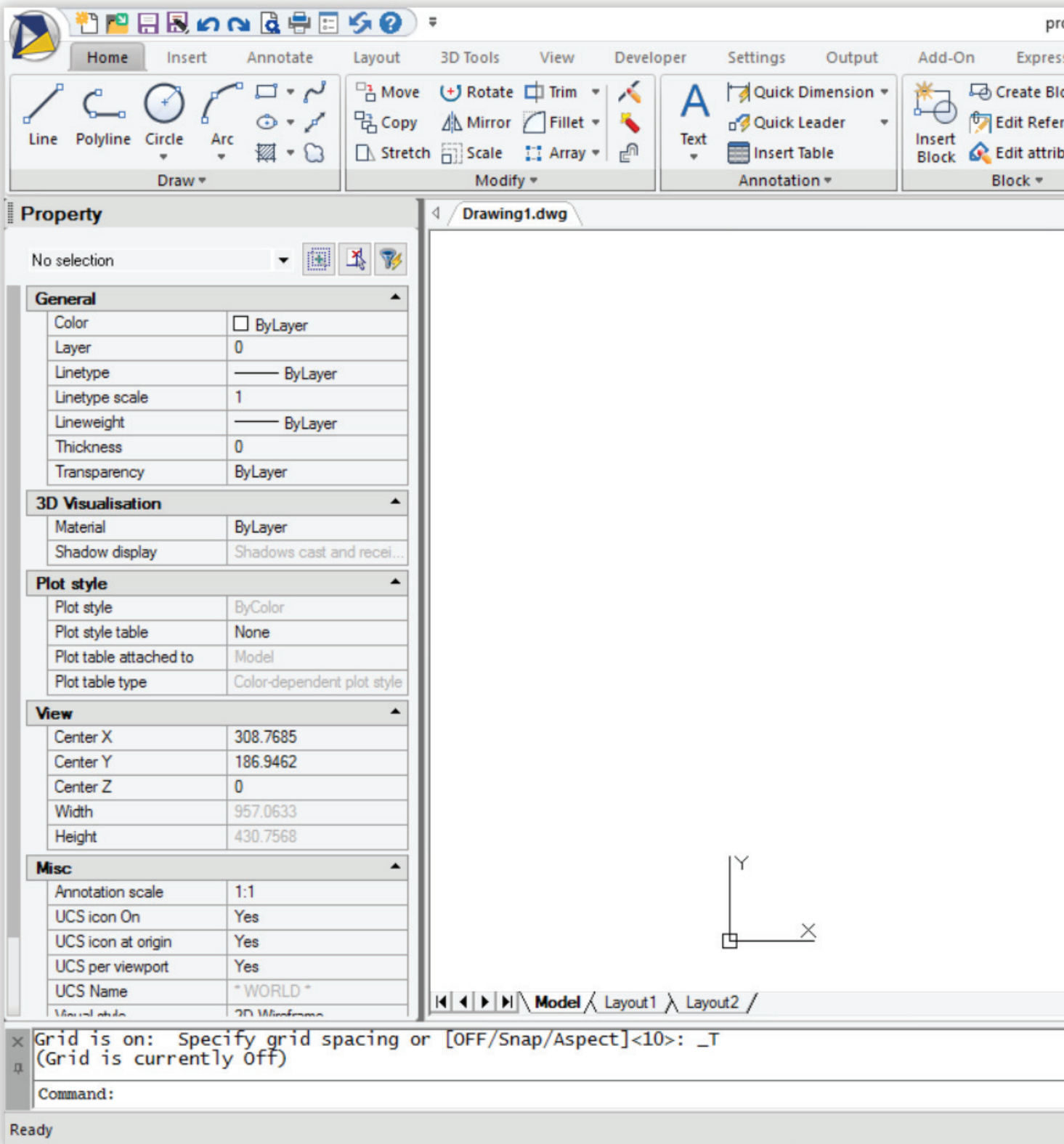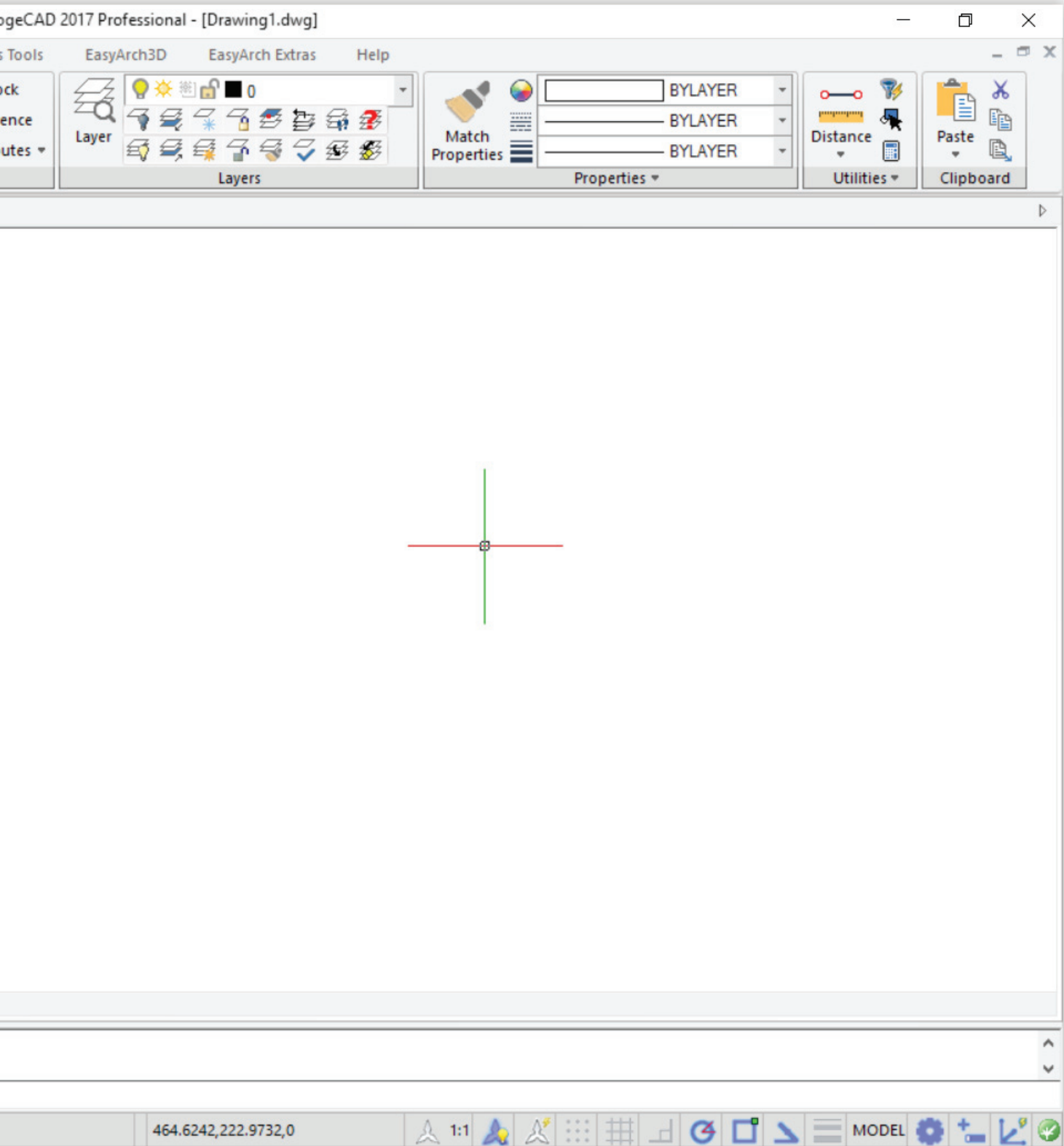
Features and Add-ons Exclusive to progeCAD

Here is a summary of features and enhancements new to the latest release of progeCAD:

DWG 2017 and Conversion

Commands like **Open** and **Save** now support drawing files right up to DWG 2017 format.

The new **Convert** command exchanges files between DWG, DXF, and DGN-based CAD software.

Ribbon User Interface

New to progeCAD 2017 is the ribbon user interface (see figure below). Customers can choose between it and the common menu-toolbar interface.
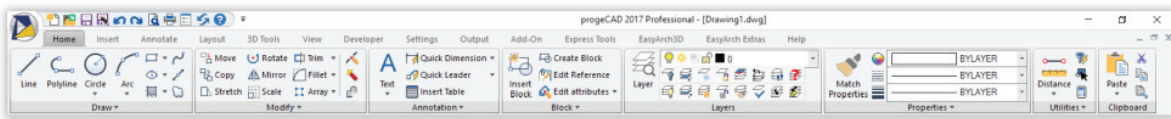
64-bit and Multi-core Support

progeCAD 2017 runs faster, because it now is available in a 64-bit versions for access to more memory, as well as multi-core support for faster processing of certain operations.

PointCloud Import

progeCAD 2017 imports and displays 3D data from point clouds produced by laser scanners (see figure below). The new **PointCloudAttach** command imports files in *.pcg* and *.isd* formats.
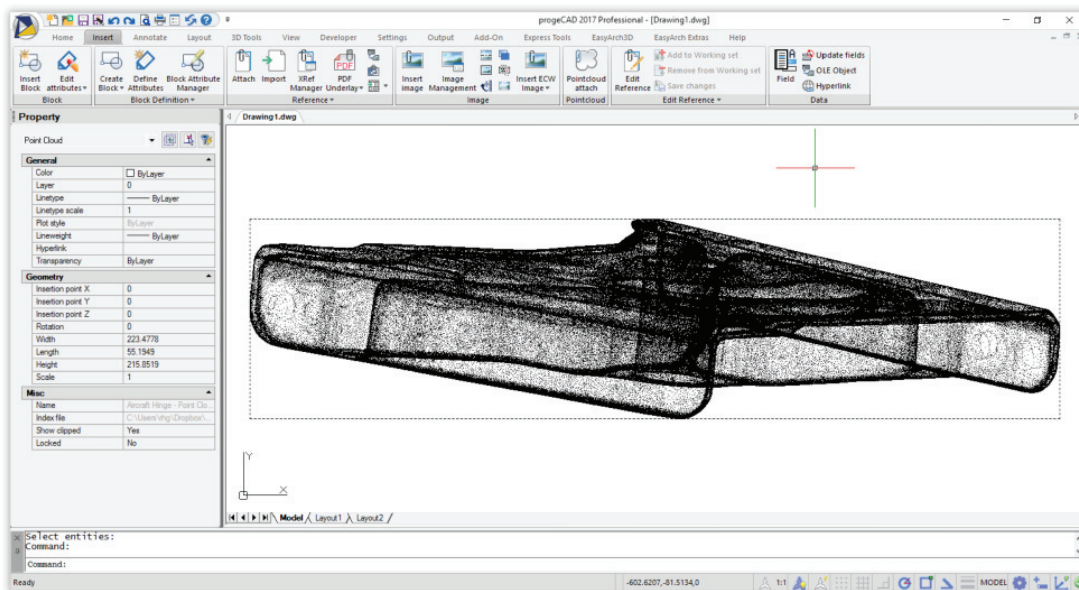
New and Improved Grips

Polylines gain new grips that stretch, add vertices, and convert segment into arcs.
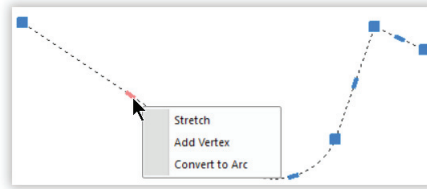
The grips in dynamic blocks are improved through extended support for dynamic blocks editing options.

Table Styles

progeCAD formats tables easily through styles using the new **TableStyle** command (see figure below).
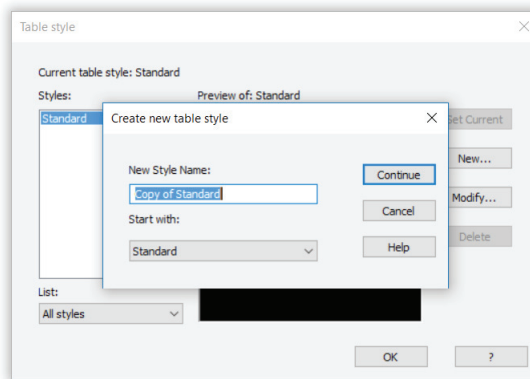
RGB and Index Colors

All color-related system variables now support RGB (red, green, blue) and index color values.

Print to DWF/DWFx files

Export drawings to the standard DWF and DWFx formats using PC3 printers and the **Print** command.

New and Improved File Interchange

progeCAD now imports Collada (*.dae*) files, and improves the way it handles DGN overlays, Civil3D files, ADT files, and mechanical object enablers. For instance, it now converts MicroStation *.dgn* to DWG format. MrSID *.mg4* compressed raster images can be attached as underlays.

New API

The AutoCAD-compatible .NET API (application programming language) supports user inter-actions in drawings.

Licensing Options

progeCAD is sold to you with a perpetual license, meaning you make a one time payment and then can continue to use it without any more fees for continued use. In addition to this basic form of license, we can provide you with flexible licensing alternatives in any of the following combinations:

> **Single Plus** allows the installation of the progeCAD software on two computers, such as a desktop and a laptop, used by a single person but not at the same time

> **USB** lets you install progeCAD on as many computers as you like, and then plug in our USB dongle to authorize the license on one computer

> **NLM** (network licence manager) floats the network license around a local area net-work, or across the internet, allowing as many computers to use progeCAD simulta-neously as the license allows

> **Site** allows you to install progeCAD on any computer at a single location, or on any computer in a certain country, or in multiple countries

Summary of New Commands

AlignSpace matches the viewpoint of a layout's viewport to points the user specifies in model and paper space

BCount counts the number of blocks in a selection set or in the drawing

Convert converts files between DWG, DXF, and DGN formats

LayerWalk displays or hides the content of selected layers

PointCloudAttach attaches point cloud files to the drawing

PolySolid draws 3D polyline-like solids with width and height

SelectSimilar selects objects with similar properties

VpSync synchronizes viewpoints between viewports

XOpen opens xrefs (externally-references drawings) for editing

Xplode explodes complex entities into simple ones, with control over the process

progeCAD is a product made by ProgeSOFT SA (http://www.progesoft.com) and is distributed in the United States by progeCAD USA (http://www.progecad.us).In addition, progeCAD USA offers industry-specific add-ons that run within progeCAD:

- progeBILLD for AEC MEP, which includes modules for electrical, mechanical, structural, plumbing, and HVAC
- progeEARTH for civil/survey work
- progeTABLE for interfacing with spreadsheet files

About This eBook

Inside progeCAD is based on progeCAD Professional 2017, a commercial version of progeCAD.

The book is divided into three major sections:

Section 1 introduces progeCAD by taking you on a guided tour. If you know AutoCAD, then you are probably already familiar with progeCAD. Feel free to skip ahead to Chapter 4, "progeCAD for AutoCAD Users" for an overview of the differences.

Section 2 takes you through a tutorial that designs the floor plan of a one-bedroom apartment.

Section 3 consists of an appendix that summarizes user interface elements.

Your Guided Tour Through progeCAD

In this chapter, you learn how to start progeCAD, and then take a guided tour of the CAD program. In addition, you learn the basics of working with a CAD system.

Starting progeCAD

You can start progeCAD in any of these ways:

- Double-click the progeCAD icon, if it is found on your computer's desktop.
- On the Windows taskbar, click the **Start** button, and then start typing "progecad." Select **ProgeCAD 2017** from the resulting list.
- In Windows Explorer or Mac Finder, double-click the name of a *.dwg* file. (This option works only when progeCAD is specified to work with *.dwg* files.)

In This Chapter

- Starting progeCAD
- Ribbon
- Workspaces
- Drawing Area
- Layout Tabs
- Prompt History Window
- Keyboard
- Coordinate Input
- Point Filters
- Dynamic Input
- Select Set Modes

- Tip of the Day
- Quick Access Toolbar
- Menu Bar
- Cursors
- Prompt Boxes
- Scroll Bars
- Shortcut Keys
- Vector Entities
- Direct Distance Entry
- Selecting Entities
- Selecting by Properties

- Title Bar
- Application Menu
- Toolbars
- UCS Icons
- Command Bar
- Status Bar
- Mouse and Tablet
- Keyboard Input
- Mouse Input
- Grips Editing
- Quick Select

Exploring the User Interface

When progeCAD starts, it presents itself like the image shown below. (Shown is the user interface for progeCAD 2017 Professional for Windows. If you use another version of progeCAD, you may see something different due to differences among user interfaces.)
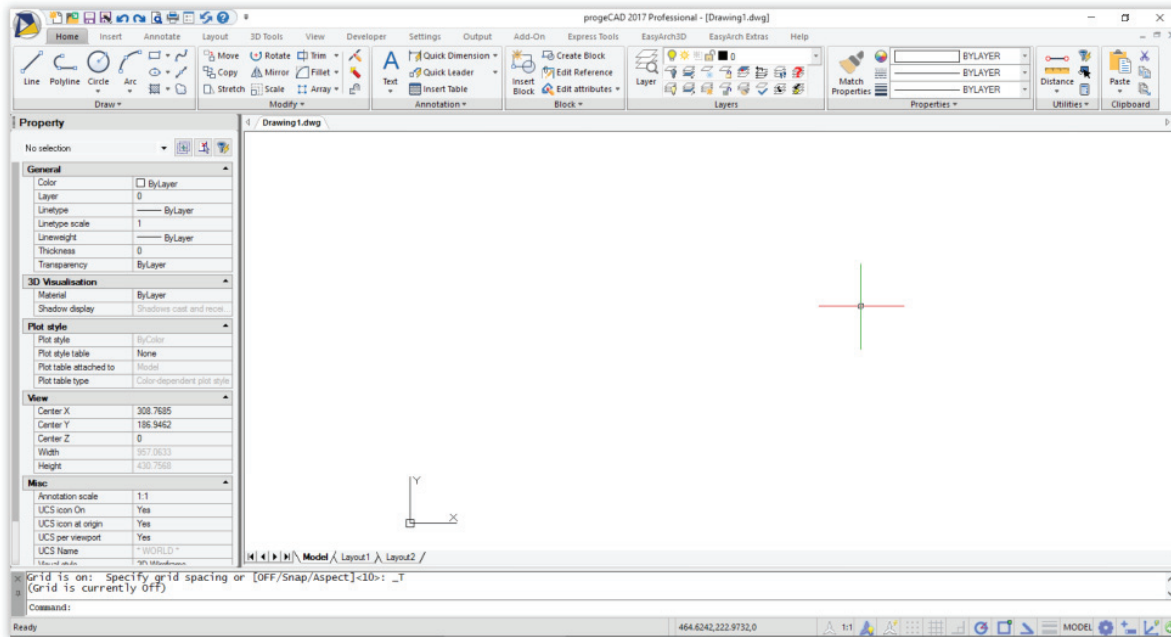
Let's take a walk through the user interface, starting from the top of the progeCAD window.

Title Bar

At the top of every program's window is the *title bar*.

It reports the name of the software (**progeCAD 2017 Professional** in this case) and the name of the current drawing: **[Drawing1.dwg]**, as in the figure above.

When drawings are opened in *read-only* mode, the phrase **(Read-Only)** is added to the file name. It reminds you that changes made to the drawing will not be saved to the same file.

To save the changes made to a read-only drawing, save it by another file name with the **SaveAs** command.

Window Controls

At the right end of the title bar is a set of three buttons that minimize (_) , restore or windowize (⬜), and close (**X**) the window. There is one set for the program, and another for the window holding the drawing.

Switching Between User Interfaces

At this point in the chapter, my description of the user interface splits into two styles, ribbon or menu-toolbars. The ribbon interface is described below, because it is the default one; the menu-toolbars interface is described in the boxed text.

When the program changes to the Ribbon menu interface, it gets three elements that are different: an application menu, a quick access menu, and a ribbon.

Application Menu

To far left of the title bar we see the Application menu ( ) that holds file and utility functions. To access these functions, click the large icon.
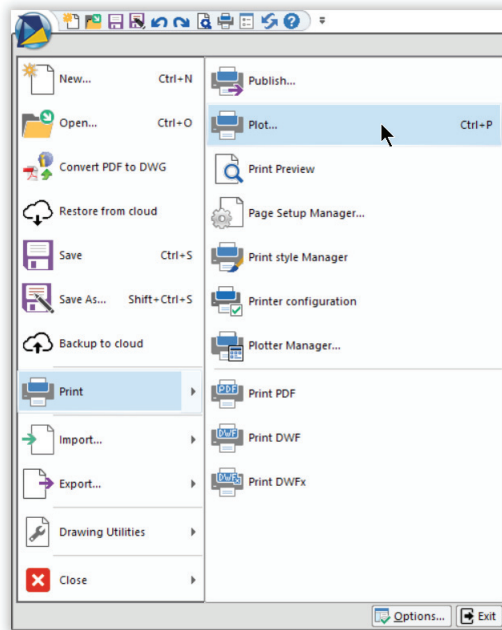
Quick Access Toolbar

Next door to the application menu is the quick access toolbar. This mini toolbar gives us access the commonly used utility functions, like saving and printing drawings.

progeCAD's User Interface

pro

| Home | Insert | Annotate | Layout | 3D Tools | View | Developer | Settings | Output | Add-On | Express |

Draw
Line Polyline Circle Arc

Modify
Move Rotate Trim
Copy Mirror Fillet
Stretch Scale Array

Annotation
Text
Quick Dimension
Quick Leader
Insert Table

Block
Insert Block
Create Blo
Edit Refer
Edit attrib

Property

◁ Drawing1.dwg Drawing2.dwg **Drawing3.dwg** 1343939490.dwg 1344465784.d

Draw

No selection

Properties palette

General

Color	☐ ByLayer
Layer	0
Linetype	——— ByLayer
Linetype scale	1
Lineweight	——— ByLayer
Thickness	0
Transparency	ByLayer

3D Visualisation

| Material | ByLayer |
| Shadow display | Shadows cast and recei... |

Plot style

Plot style	ByColor
Plot style table	None
Plot table attached to	Model
Plot table type	Color-dependent plot style

View

Center X	397.6611
Center Y	235.05
Center Z	0
Width	357.6323
Height	123.9508

Misc

Annotation scale	1:1
UCS icon On	Yes
UCS icon at origin	Yes
UCS per viewport	Yes

Drawing area

UCS icon

Origin of the drawing (0,0)

Layout tabs

|◁ ◁ ▷ ▷|\ **Model** ⟨ Layout1 ⟩ Layout2 /

```
Entities in set: 2
Select entities to display grips or [FILter/Qselect]:
Command: _DELETE
2 found
Command: _PROPERTIES
Command:
```

Displays objects on layers that you select in the Layer list.

Command prompt

progeCAD 2017 Professional - [Drawing3.dwg]

Tools   EasyArch3D   EasyArch Extras   Help

Layers

Layer

BYLAYER
BYLAYER
BYLAYER

Match
Properties

Properties ▾

Distance

Paste

Utilities ▾   **Clipboard**

...wg (Read-only)   Drawing4.dwg

Crosshair cursor
(green = y
red = x)

Pick box
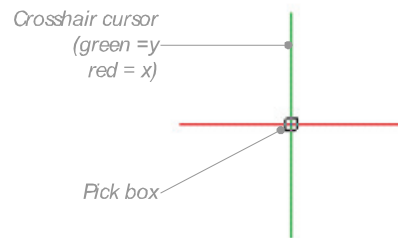
Vertical scroll bar

Horizontal scroll bar

106.3995,-24.0957,0   1:1   MODEL

X, y coordinates and
elevation

Status bar toggles

From left to right, the icons represent commands:

- Starting new drawings
- Opening drawings
- Saving the current drawing
- Saving the drawing by a different name or to a different folder
- Undoing the last command
- Redoing the last undone command
- Previewing what a print of the current drawing looks like
- Printing the current drawing
- Toggling (turning on and off) the Properties palette
- Regenerating (refreshing) the current drawing
- Accessing help

You can change which buttons this toolbar displays by clicking the down arrow ( ) and choosing commands from the list.
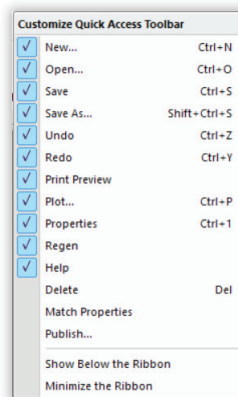
The check marks indicate which commands are displayed by the quick access toolbar. Click a command name to turn the check mark on or off.

At the bottom of the droplist are two options:

Show Below the Ribbon — when this option is turned on, the program relocates the quick access toolbar from the title bar to just below the ribbon
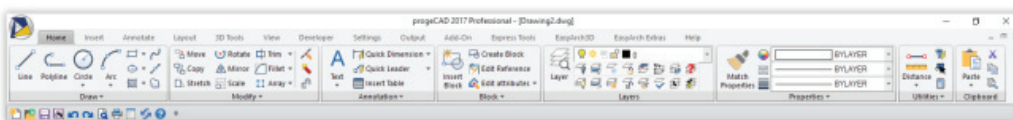
Minimize the Ribbon — when this option is turned on, the program reduces the height of the ribbon so that just the tab names are shown
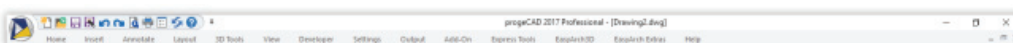
You still can access the buttons of the ribbon by clicking on the name of a tab.

Inside progeCAD 2017

Ribbon

Below the title bar is the ribbon. It's kind of operates like overlapping toolbars, where of tabs segregate the "toolbars" into groups of functions.

Tab names

Panel names
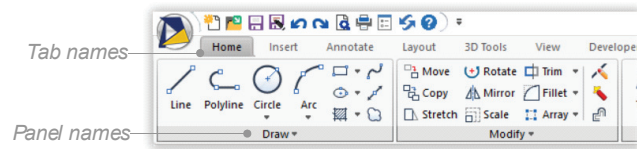
Tabs group similar functions, with names such as Insert (inserting objects) and Annotate (placing text). *Panels* segregate tabs into groups of related commands, such as Draw (drawing entities) and Modify (editing the entities). Panels can contain groups of buttons, flyouts, and/ or droplists — just like toolbars. You can change content of the ribbon with the **Customize** command.

TIP   The ribbon can be turned on and off with the **Ribbon** variable:
: **ribbon**
New value for RIBBON <On>: **off**

Menus and Toolbars

To switch between user interface styles, right-click the workspace button on the status bar: **Workspace** command like this:
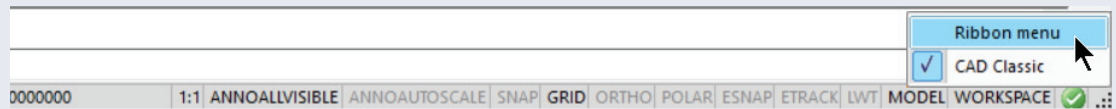
"Ribbon menu" switches to the ribbon interface

"CAD Classic" switches to the menu and toolbars interface

Menu Bar

The *menu bar* is located below progeCAD's title bar. Its purpose is to sort progeCAD's many commands into logical groups. For example, almost all file-related commands, such as Open, Save, and Close, are found in the File menu. All commands related to dimensions are in the Dimensions menu.
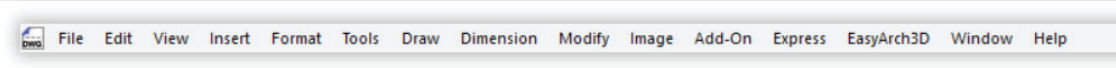
progeCAD's menus operate like those in other Windows programs. Select a menu item, and then select a command from the dropdown menu. For example, to save the drawing, use the mouse and cursor to select the **File** item, and then choose **Save**.
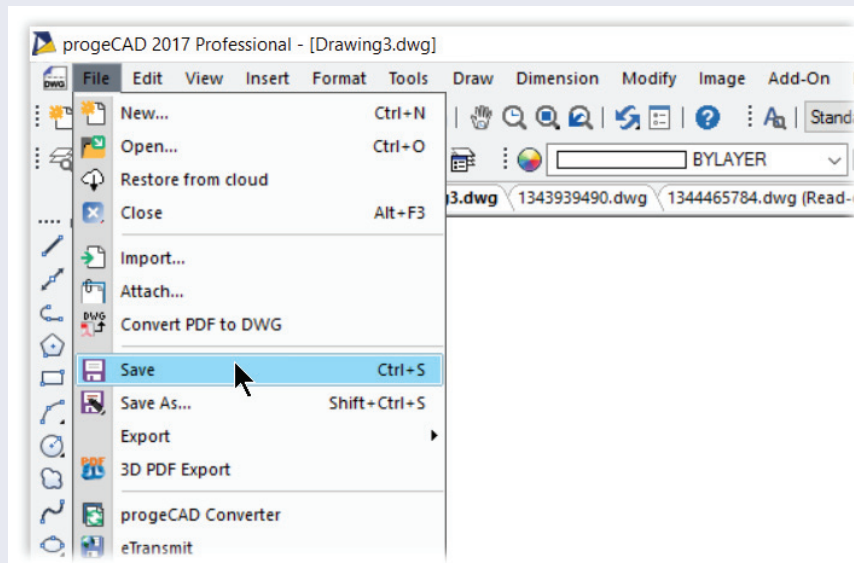
Menus use these notations to guide you as to the item's purpose:

File (underlined letter) means that the menu item can be accessed from the keyboard, in place of the mouse. To do so, hold down the **Alt** key, and then press the letter — **F** in this case.

Ctrl+N means the command can be activated with a keyboard shortcut. In this case, hold down the **Ctrl** key, and then press the **N** key to activate the New command. Other keyboard shortcuts involve the **Alt**, **Shift**, and F (function) keys.

New**...** (ellipsis or three dots) means the command displays a dialog box. When there are no ellipses, look for prompts on the command line or in the prompt box.

_____ (separator line) separates groups of commands within the dropdown menu.

✔ (check mark) means the option is turned on; no check mark means the option is turned off. You turn the option on and off by selecting the menu item.

▸ (arrowhead) means the menu item displays an additional menu, also called a "sub menu." In some cases, sub menus display further sub menus.

You customize the menu bar and dropdown menus with the **Customize** command.

Toolbars

While menus display commands as words, toolbars display commands as pictures on buttons called "icons." As in other Windows programs, you activate a command by clicking its button.
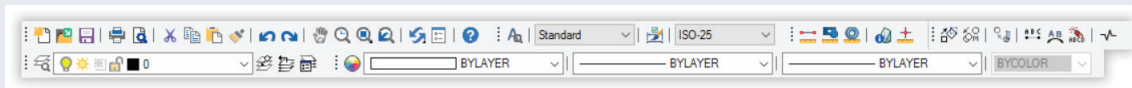
Being pictures, icons are not always clear as to their purpose, and so progeCAD provides assistance in decoding their meaning.

When you move the cursor over a button, look to the status line at the bottom of the window: it displays a one-sentence description of the button's purpose.

When you leave the cursor sitting over the toolbar button, a *tooltip* appears after a few seconds. The tooltip's rectangle contains a word or two describing the command executed by pressing the button.
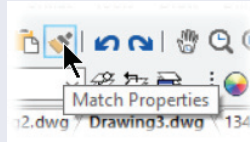
progeCAD displays a number of toolbars initially,but it has many others. To display them, right-click any toolbar, select **PCAD**, and then choose a name from the list.

On high resolution monitors, the buttons might be too small to see. You can double their size by right-clicking any toolbar, and then select **Toolbars**. Select **Large Buttons**, and then click **OK**.
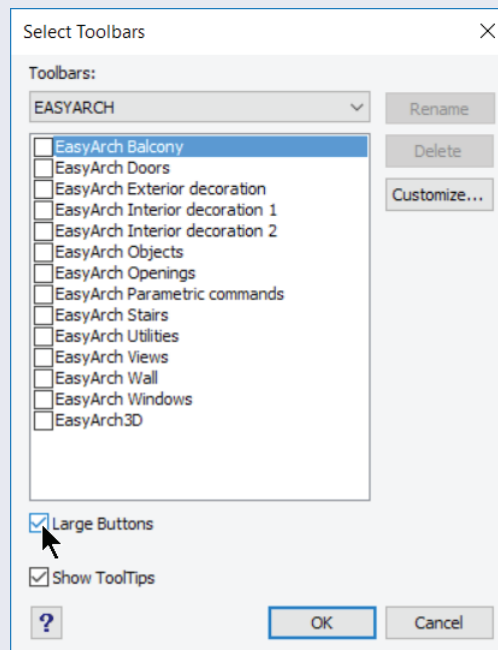
You can change any toolbar and buttons with the **Customize** command.

Entity Properties Bar

The Entity Properties bar is a special toolbar that shows you the settings for (left to right) colors, linetypes, lineweights, and (if enabled) plot styles.
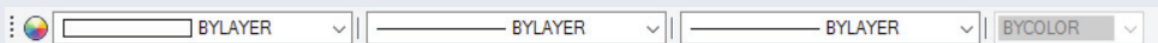
It works in two ways:

- When no objects selected, the drawing's default settings are reported, as shown above.

- One or more objects are selected, it reports settings for the selected object(s), and lets you modify the properties of the selected object(s).

Drawing Tabs

progeCAD can display one or more drawings at a time, each in its own window. To open more than one drawing, just enter the **Open** command; the drawing file you select is opened in a new window.

To switch between drawings quickly, progeCAD sports a row of tabs between the ribbon (or toolbars) and the drawing area. For every drawing that is currently open in progeCAD, its file name appears on a tab. Click a tab to switch to the associated drawing.

Alternatively, you can switch between drawings with the Windows menu, or by pressing **Ctrl+Tab**. Right-click a drawing tab to see a shortcut menu of options:
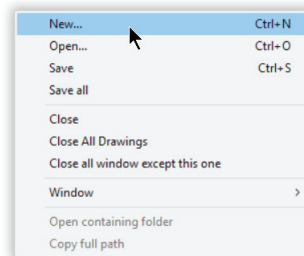
This menu has useful options not found elsewhere in the program:

> **Save All** saves all the drawings at once
>
> **Close All Drawings Except This One** closes all drawings, except the current one
>
> **Open Containing Folder** opens the file manager at the folder in which the *.dwg* file is stored
>
> **Copy Full Path** copies the full path of the *.dwg* file to the Clipboard, including the drive name; the path can be pasted into another program, such as a text editor or the address bar of a file manager.

The last two options are not available to new drawings that are not yet saved or were created from a template, because they do not have a folder assigned to them until they are saved.

Drawing Area

The drawing area is where you view, draft, and edit drawings.

When you start a new drawing with the **New** command, progeCAD opens a new, blank drawing. When you open existing drawings with the **Open** command, they appear in the drawing area.

You can change the color of the drawing area. Some drafters like it to be black, because it makes entity colors stand out. Other drafters (myself included) prefer white for the background color, because this color is the same as that of the paper upon which drawings will be printed.

You can change the background to any of 254 colors and shades of gray, as follows:

1. Enter the **Options** command. Notice the Options dialog box.
2. Select the **Display** tab, and then click the **Color Schemes** button.
3. Under Items, choose **Background**, and then choose a color from the **Color** drop list.
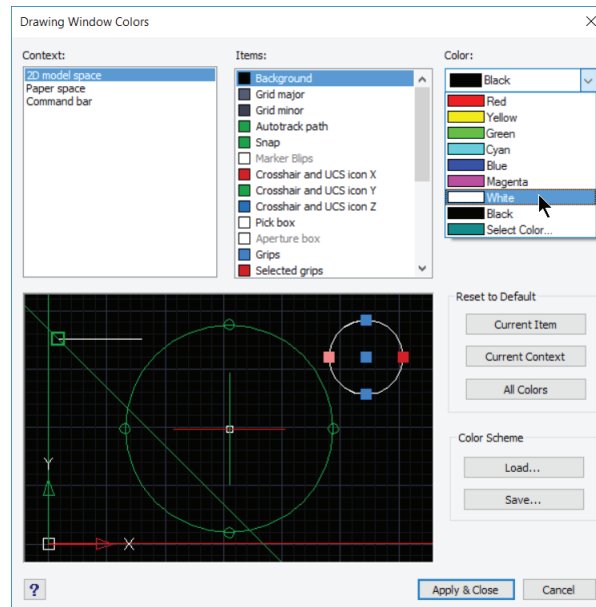
4. Click **OK** twice to dismiss both dialog boxes.

(This book illustrates progeCAD with the drawing area white.)

Cursors

The cursor shows you where "you" are in progeCAD. It also provides you with information: in different areas and in different situations, it changes its shape.

Outside of the drawing area, the cursor displays the familiar arrowhead found in other Windows applications.

When the cursor is paused over a toolbar button, a *tooltip* appears after a second or two. The tooltip explains the purpose of the button. If you do not see tooltips, you can turn them on: right-click any toolbar, and then select **Toolbars** from the shortcut menu; choose the **Show Tooltips** option.

On the edge of windows, toolbars, and stretchable dialog boxes, the stretch cursor appears, indicating you can change the size of the user interface element.

In the drawing area, the cursor changes in these ways:

Cursor   Meaning

No Command Active: progeCAD is ready to select an object. The small square is called the "pickbox." Within it, progeCAD searches for objects. The size of the pickbox is changed with the **Pickbox** system variable.

The colored cursor indicates the orientation of the x (red) and y (green) axes. In 3D space, the z axis is indicated by a blue cursor line. The colors of the cursor are changed with the **Option** command's **Cursor** option.

Command and Esnap (entity snap) Active: progeCAD is ready to select objects. The large square is called the "aperture." It is displayed only when esnaps are turned on. progeCAD searches for geometric features within the aperture. The color and size of the square are changed with the **Options | Snapping** command's **Display Snap Aperture Box** option.

The "drumstick hitting the bassdrum" cursor is called the "snap cursor decoration." It alerts you that two or more entity snaps are active. Other decorations are displayed for specific entity snaps. (See full list later.) Decorations are off by default; to turn them on, use the **Options | Snapping** command's **Show Snap Setting on Cursor** option.

Entity Snap Found: When progeCAD finds a geometric feature, it displays a green icon called the "marker," which shows the location of the geometric feature; a tooltip describes the feature, "Endpoint", in this case. (See the full list below.) The color and size of the marker are changed with the **Options | Snapping** command's **Snap Marker** option.

Entity snaps employ two types of icons: *decoration* icons that report the current entity snap mode, and *marker* icons that report when progeCAD finds the matching geometric feature. Here is the list of entity snap icons and their meanings:

Esnap Mode	**Decoration**	**Geometric Marker**
Endpoint		
Nearest		
Perpendicular		
Quadrant		
Parallel	*no icon*	
Center		
Midpoint		
Node		
Tangent		
Intersection		
Apparent Intersection	*no icon*	
Insertion		
Extension		

Zoom. Magnifying glass icon is displayed during the Zoom command.

Pan. Hand icon is displayed during the Pan command.

Real Time Rotation. 3D spin icon is displayed during the RtRot command. You will see these icons during modes of real time view rotation:

RtRotX. Spin about x axis.

RtRotY. Spin about y axis.

RtRotZ. Spin about z axis.

UCS Icon

The purpose of the *UCS icon* is to show you the drawing's orientation in 3D space. UCS is short for "user coordinate system." The icon is usually found in the lower left corner of the drawing area.

In 2D space, the x and y axes are shown; the z axis is hidden because in this mode we are looking straight down on the x,y-plane. When the letter W appears, it means that the view is in the "world coordinate system" — basically, this is the default and any time a UCS is not invoked. (See figure below, at left.)
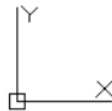
In 3D space, the z axis shows up.
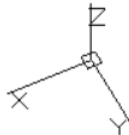
The UCS icon is pointless for most forms of 2D drafting, and so can be turned off with the **Uc-sIcon** command's **Off** option. The sole exception is when you need to draft at an angle, such as the wing of a building that is, say, 22.5 degrees to the main building.

When in paper space (layout mode), the UCS is also pointless, because paper space is always 2D-only. (Paper space is discussed in a later chapter.) Nevertheless, the UCS icon appears there as an icon that mimics a drafter's triangle.

Layout Tabs

Layout tabs are located below the drawing area. They allow you to switch quickly between model space and named layouts. *Model space* is where the drawing is created, where the 2D or 3D model is designed. *Layouts* are meant primarily for plotting: you arrange 2D views of the model inside viewports with drawing borders or title blocks.

There is just one model tab, but one or more layouts. You would work with more than one layout when you want to plot the drawing on different sizes of paper or on color and black-white printers. Multiple layouts allow you to arrange the drawing in a variety of ways before plotting.

When you right-click a layout tab, a shortcut menu displays options for creating, renaming, and deleting these tabs.
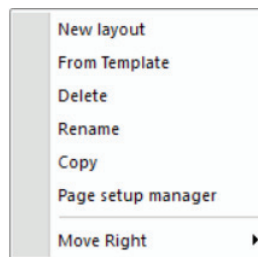
The Model tab cannot be renamed nor erased, and the first layout tab cannot be erased.

Scroll Bars

In the same area as the layout tabs is the horizontal scroll bar. It lets you pan the drawing left and right; a second vertical scroll bar lets you pan up and down.

I find scroll bars very handy, and so I always turn them on. In the Options dialog box, click the Display tab, and then choose the **Show Scroll Bars** option. Or, you can enter the **ScrollBar** command.

Command Bar

The *command bar* is the area where you enter commands; this is also the area where progeCAD most often reports information to you. Typically, you look here to see command options; this does not apply to dialog boxes, 'natch.
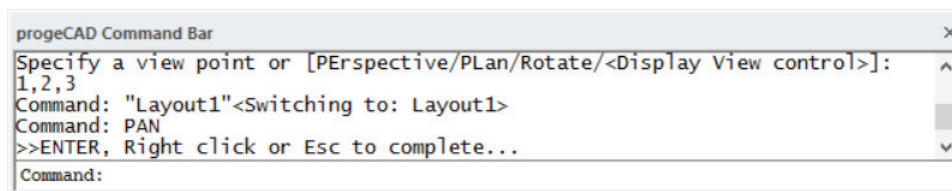
The command bar normally hangs off the bottom of the progeCAD window, but you can drag it away to make it a floating window. Once floating, you can resize the command bar to be wider, narrower, taller, and shorter.

I find no benefit, however, to floating the bar,and so I keep it firmly attached to progeCAD. If I need to see more of what is going on, then I use the prompt history window, described next.

You can double-click the status bar to toggle the display of the command bar. Double-click once, and it turns off; double-click a second time, and it turns back on. (*Toggle* means to turn on and off.)

Command Prompt Options

progeCAD offers you a variety of ways to see and enter command names. This is done through a shortcut menu that you see when you right-click the command bar:
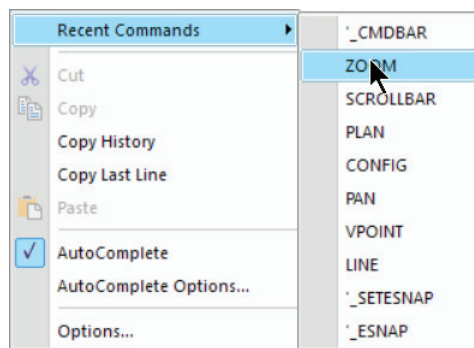
Recent Commands lists the last ten commands you entered. A better way to access recently-entered commands is to just press the up arrow on the cursor pad, I find.

Cut and **Copy** cut and copy text (that you highlighted) and places it in the Clipboard; the text can be pasted back into the command bar, or into another document.

Copy History copies all the text in the Prompt History window, and places it in the Clipboard. This can be handy for creating script files or documenting steps for creating drawings.

Copy Last Line copies the last line of command text to the Clipboard.

Paste pastes text from the Clipboard into the command prompt; this works only when the Clipboard contains text. This is handy for repeating a complex command or executing LISP code.

AutoComplete lets the program guess which command you want to enter, and so presents you with a list of all commands that start with the same few letters.
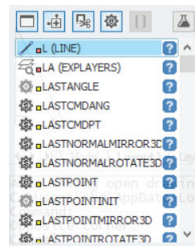
Click the blue ? for help about the command. The buttons along the top toggle options — commands, external commands, aliases, system variables — or access the options shown below.

AutoComplete Options displays a dialog box that lets you control how auto complete works.
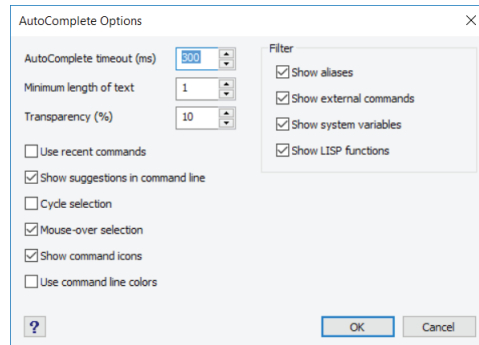
Options displays the Options dialog box.

Command Prompt

The command *prompt* is the word 'Command' followed by the colon, like so:

    Command:

When you see the prompt by itself on the lowermost line of the command bar, you know that progeCAD is ready for you to enter a command. If it is not clear, press **Esc** once or twice to clear the prompt line.

TIP   Use the **CmdLnText** system variable (short for "command line text") to change the prompt wording from 'Command:' to something else, should you wish.

Prompt History Window

Press function key **F2** to see the *prompt history window*. The window remembers the last several hundred lines of commands and prompts.
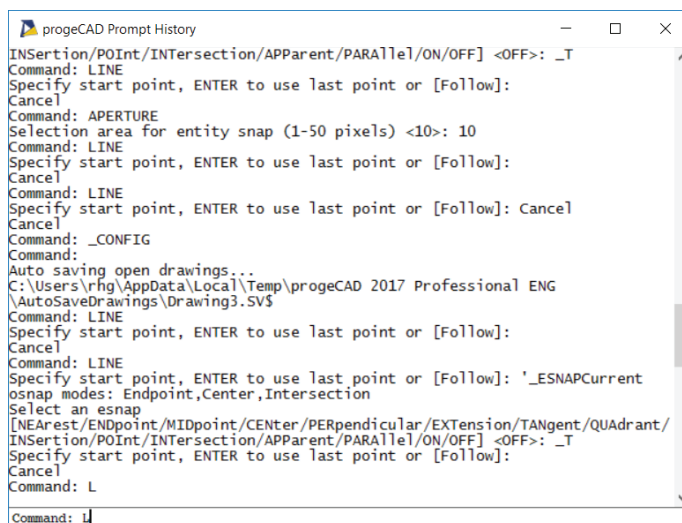
Right-click the prompt history window to display a shortcut menu that lets you copy the text to the Clipboard, or to paste text into the window.
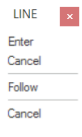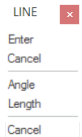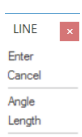
You can change the number of lines remembered by progeCAD: enter the **Options** command, click the **Display** tab, and then modify the value of **Command Lines to Track**. The maximum and minimum values are 500 and 1 lines.

Prompt Boxes

Many of progeCAD commands involve *options*. The **Line** command, for example, has options for undoing (un-drawing) the last segment, closing the lines as a polygon, and so on. Typically, you specify the option by entering its first letter, such as "U" for Undo and "C" for Close.

progeCAD provides an alternative to typing characters: *prompt boxes* are shortcut menus that display lists of option names, which you can select with the cursor. The display of prompt boxes can be turned on and off through the **Options** command's Display tab and Menus section: change the setting of **Display prompt boxes**.

Here is an example of the different prompt boxes that appear during the **Line** command:

(no prompt box) Command: **line**

(no prompt box) Start of line: *(Pick a point.)*

LINE ✕
Enter
Cancel
Follow
Cancel

Next point or [Angle/Length]: *(Pick another point, or select an option.)*

LINE ✕
Enter
Cancel
Angle
Length
Cancel

Next point or [Angle/Length/Follow/Undo]: *(Pick another point, or select option.)*

LINE ✕
Enter
Cancel
Angle
Length
Follow
Undo
Done
Cancel

Next point or [Angle/Length/Follow/Close/Undo]: *(Pick a point, or select option.)*

Prompt boxes disappear when commands end. Some commands, such as **Circle** and **Move**, end on their own; for other commands, such **Line** and **Trim**, you need to press **Esc** or **Enter** to end them.

Status Bar

The status bar reports information on the status of the drawing and the drafting process. From left to right, the bar provides the following info:

| Measures the distance and angle between two points: DIST | 140.5362,117.2513,0 | | | P:Layout1 |
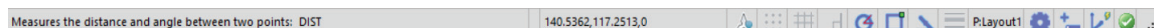
Help text is a sentence that appears when selecting menu items and toolbar buttons; otherwise, the word "Ready" appears. This part of the status bar performs a couple of "hidden" functions:

- Right-click this part of the status bar to control which buttons are displayed. I like turning on the **Show Text** option, because frankly icons don't mean a lot to me.
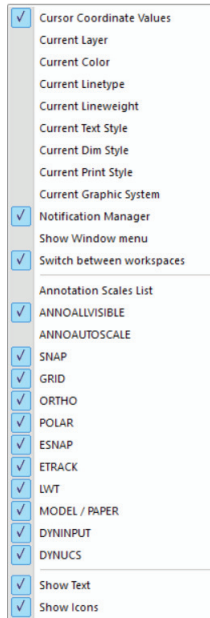
- Double-click this part of the status bar to toggle the display of the command bar.

X,y,z reports the coordinates of the cursor position in the drawing area; the z coordinate reports the current elevation setting.

During drawing and editing commands, the display changes to show the length (distance) and angle from the point you last picked in the drawing.

Right-click the coordinates to change the display among the following units:
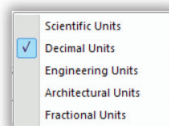
- Scientific with exponents
- Decimal, the default, also used for metric drawings
- Engineering with feet and decimal inches
- Architectural with feet and fractional inches
- Fractional with only inches, no feet

Here I list the default items displayed on the status bar. Right-clicking most of the buttons accesses the Settings dialog box, which affects how the functions look and operate.

- **1:1** shows the current annotation scale; button is displayed in model mode. Right-click this button for the list of available annotation scales.
- **AnnoAllVisible** toggles the display of all scales assigned to annotative objects; button is displayed in model mode. This one is useful for debugging annotation scaling.
- **AnnoAutoScale** toggles whether newly selected annotation scales are attached to annotative objects; button is displayed in model mode. I find it is best to leave this option off.
- **Snap** toggles snap mode.
- **Grid** toggles the display of the grid.
- **Ortho** toggles orthographic mode, which constrains cursor movement to right angles. (This is turned off automatically when Polar is turned on.)
- **Polar** snaps the cursor at preset angles, such as 15 or 30 degrees; right-click to choose a preset angle. (This is turned off automatically when Ortho is turned on.)
- **ESnap** toggles entity snap mode; right-click to access the list of all esnap modes.
- **ETrack** toggles entity tracking mode.
- **LWT** toggles the display of lineweights.
- **Model** toggles the viewport between model and paper space (**P:Layout**).
- **Workspace** changes workspaces.
- **DynInput** toggles the display of dynamic input; discussed later in this chapter.
- **DynUCS** toggles the use of dynamic UCS, where the orientation of the UCS plane automatically matches the feature on the currently-edited 3D entity.
- **Events Log** displays a dialog box reporting events, such as program errors.
- *Corner* makes the progeCAD window larger and smaller, when dragged with the cursor.

To change the setting of a toggle, click its "button." When on, the icons look blue and the words (SNAP, GRID, and so on) look black; when off, the icons and words are gray.

Toggling the Status Bar

The display of the status bar can be turned on and off with the **StatBar** command. You might want to turn it off to gain a fraction of an inch more drawing area, but the status bar is far too useful to keep off. It provides an overview at the state of the drawing, and provides instant access to frequently-used toggle commands.

User Input

progeCAD accepts input from the keyboard and mouse. It also supports other input devices, such as touch screens and 3D mice — provided drivers are available that are recognized by progeCAD.

Keyboard

It is possible to run progeCAD *almost* entirely by mouse; the keyboard is still needed for a few input tasks, such as certain coordinates, entering certain command options, and the naming of files. Indeed, there are some users — such as me — who prefer the keyboard over a mouse.

Commands and options are typically entered by typing their names in the command bar.

```
progeCAD Command Bar                                                    ×
Specify a view point or [PErspective/PLan/Rotate/<Display View control>]: ∧
1,2,3
Command: "Layout1"<Switching to: Layout1>
Command: PAN
>>ENTER, Right click or Esc to complete...                              ∨
Command:
```
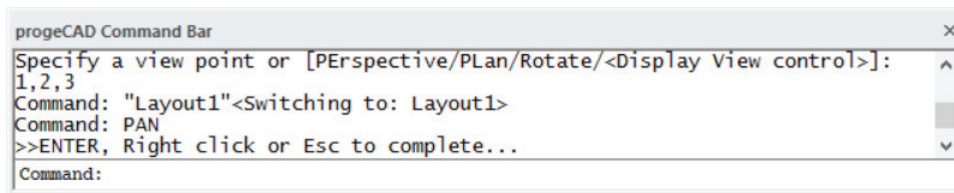
What the Space Bar and Enter Do

Pressing the *space bar* performs several functions, whose meaning depends on *when* you press it: end input, exit commands, or repeat commands. Pressing the space bar is equivalent to pressing the **Enter** key.

The space bar tells progeCAD that you are done entering data, such as the name of a command, an option, or coordinates:

> Command: **line** *(Press spacebar or **Enter**.)*
> Start of line: *(Type coordinates, and then press spacebar or **Enter**.)*

Press the spacebar to tell progeCAD you want the command to end:

> Next point or [Angle/Length]: *(Press spacebar or **Enter**.)*
> Command:

At the **:** prompt, the spacebar tells progeCAD to repeat the previous command:

> Command: *(Press spacebar or **Enter**.)*
> LINE
> Specify start point, ENTER to use last point or [Follow]:

The Esc Key

Pressing the **Esc** key means "cancel." You press it to get out of commands and to cancel grips.

Sometimes you have to press **Esc** two or three times to get out of deeply nested commands, like **PEdit**. It is also the equivalent of clicking the **Cancel** button in dialog boxes.

Cursor Keys

The keys found on the numeric keypad control panning. (If your keyboard lacks the extra keypad, then this features won't work.) *Panning* moves the drawing inside the window; it is most useful when zoomed in close. Press the cursor keys (up, down, left, right) to move the drawing a small amount.

To pan by an entire screen full, hold down the **Shift** key while pressing the cursor keys.

Shortcut Keys

Shortcut keys let you carry out commands by pressing assigned keys on the keyboard. Some users find this much faster than selecting commands from menus, or typing entire command words at the keyboard.

Out-of-the-box, progeCAD defines the shortcut keystrokes shown by the table below. These keystrokes can be changed through the **Customize** command.

Shortcut	Meaning	Command Executed
Function Keys		
F1	Displays on-line help	HELP
F2	Displays Prompt History window	PMTHIST
F3	Toggles entity snap mode	ESNAP T
Alt+F3	Closes the current drawing	CLOSE
Alt+Shift+F3	Closes all open drawings	CLOSEALL
F5	Switches to next isoplane	ISOPLANE
F6	Toggles dynamic UCS	DYNUCS T
F7	Toggles display of the grid	GRID T
F8	Toggles ortho mode	ORTHOGONAL T
Alt+F8	Runs VBA code at the command line	VBARUN
F9	Toggles snap mode	SNAP T
F10	Toggles polar tracking mode	POLARTRACK T
F11	Toggles entity tracking mode	ENTTRACK T
Alt+F10	Toggles status bar	STATBAR T
Alt+F11	Runs Visual Basic for Applications	VBA
F12	Toggles dynamic input	DYNINPUT T
Control Keys		
Ctrl+1	Toggles the Properties palette	PROPERTIES
Ctrl+2	Toggles the Layers palette	LAYER
Ctrl+9	Toggles the Command bar	CMDBAR
Ctrl+A	Selects all objects in drawing	SELGRIPS ALL
Ctrl+C	Copies selected objects to Clipboard	COPYCLIP
Ctrl+Shift+C	Copies objects with the basepoint	CopyBase
Ctrl+E	Switches to next isoplane	ISOPLANE
Ctrl+H	Backspaces the command line	...
Ctrl+I	Toggles coordinate display	COORDINATE T
Ctrl+K	Displays the previous command	...
Ctrl+L	Displays the next command	...
Ctrl+M	Same as pressing **Enter**	...
Ctrl+N	Starts a new drawing	PNEW

Ctrl+O [*]	Opens a drawing file	OPEN
Ctrl+P [*]	Prints the drawing	PRINT
Ctrl+Q	Exits the program	EXIT
Ctrl+R	Switches to the next viewport	^V
Ctrl+S [*]	Saves the drawing	QSAVE
Ctrl+Shift+S	Prompts for a file name upon saving	SAVEAS
Ctrl+T	Toggles tablet mode	TABLET T
Ctrl+V [*]	Pastes from Clipboard into drawing	PASTECLIP
Ctrl+Shift+V	Paste objects as a block	PASTEBLOCK
Ctrl+X [*]	Cuts selected objects to Clipboard	CUTCLIP
Ctrl+Y [*]	Redoes the last undo	REDO
Ctrl+Z [*]	Undoes the last command	U

Other Keys

Del [*]	Deletes selected objects	DELETE
Left Arrow	Pans left (numeric keypad only)	PAN L
Right Arrow	Pans right	PAN R
Down Arrow	Pans down	PAN D
Up Arrow	Pans up	PAN U
Shift+Left Arrow	Pans left a page full	PAN PGL
Shift+Right Arrow	Pans right a page full	PAN PGR
Shift+Down Arrow	Pans down a page full	PAN PGD
Shift+Up Arrow	Pans up a page full	PAN PGU
Page Down	Pans down a page full	PAN PGD
Page Up	Pans up a page full	PAN PGU

[*] Shortcut key is standard among Windows software.

Mouse

Like other Windows software, progeCAD supports the multi-button mouse as an input device. It does not, however, allow you to change the meaning of the buttons. The functions of mouse buttons vary, depending on the current circumstance. Here is an overview of their actions.

First (Left) Button

One click of the first (left) button always selects something — whether a menu, a toolbar button, or an entity in the drawing. You can click different parts of the status bar for different effects (see the section on the status bar, earlier.)

Right button (#2)
For accessing context menus

Middle button (#3)
and roller wheel for zooming

Left button (#1)
For selecting entities

Double-clicking the left button performs additional tasks that depend on the context. (Double-clicking means pressing the first button twice quickly.)

For instance, when you double-click...

- A single-line text entity, progeCAD displays the edit text dialog box.
- A multiline text, progeCAD displays the mline text dialog box.
- A dimension, progeCAD displays the dimension setting dialog box.
- A hatch pattern, progeCAD display the boundary hatch dialog box.
- Any other entity, progeCAD displays the Properties palette box with the properties of the entity.
- progeCAD's title bar, the window changes between maximized and windowed.

TIPS Through the mouse driver, I like to assign double-click to the middle button. That saves wear and tear on my hand, because I press the middle button once to get the double-click effect.

The **MButtonPan** system variable toggles behavior of the middle mouse button.
0	Operates as defined by the *.mnu* file.
1	Hold and drag to pan the drawing.

When you hold down the **Ctrl** key and press the first button, progeCAD goes into real-time 3D motion (same as executing the **RtRot** command).

Second (Right) Button

One click on the second (right) button repeats the last command.

Right-click a toolbar, the status bar, the command bar, or a selected entity brings up a context-sensitive shortcut menu.

Hold down the **Shift** key, and press the right button to display a shortcut menu of entity snap modes.

Hold down the **Ctrl** key, and hold down the right button for real-time rotation around the z axis (**RtRotZ**).

Roller Wheel

Rolling the roller wheel on the mouse (if it has one) pans the drawing up and down. To pan left or right and to zoom the drawing:

- Hold down the **Shift**+**Ctrl** keys together to pan left and right.
- Hold down the **Ctrl** *or* **Shift** keys to zoom the drawing in and out.

You can reverse the direction of the zoom relative to the roller wheel through the **Options** command's **Reverse mouse wheel zoom direction** option (found in the Display tab).

When you draw and edit, the program lets you work heads-up. Instead of entering commands and options at the command prompt, dynamic input lets you to see and enter command options, coordinates, distances, and angles right near to where you are drawing.
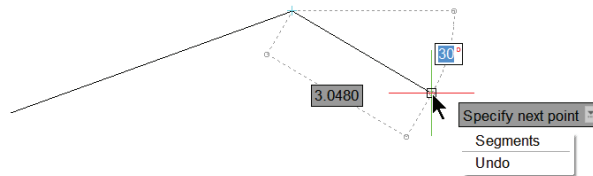
To turn dynamic input on and off, click the dynamic input button on the status bar.

Using dynamic input the first time can be tricky, so here is a tutorial on employing the heads-up interface:

1. Start the **Line** command. Notice that the command's name appears in the drawing area.
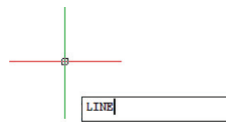
2. Press **Enter**. Notice that dynamic input shows four pieces of information. From left to right, these are as follows:

   • **Prompt** shows "Start of line."

   • **#** means that no options are available right now.

   • **X-coordinate** of the cursor location, such as 336.7 illustrated below. The coordinate value is highlighted in blue; this indicates the value can be edited.
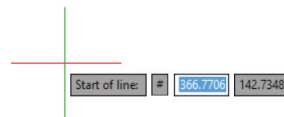
   • **Y-coordinate** of the cursor location, such as 142.7 illustrated above. The coordinate value is highlighted in gray; this indicates the value cannot be edited — for now.

3. To edit the y-coordinate, press **Tab**. Notice that the y-coordinate now turns blue, meaning you can edit it. (The x-coordinate now is gray.) You can press **Tab** repeatedly to move between the two fields.
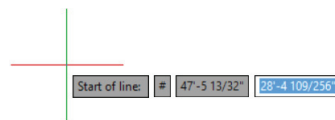
4. To place the starting point, click with the mouse or else press **Enter** to accept the x and y values you entered.

5. Move the cursor. As you do, more dynamic input elements appear. From top to bottom, they are as follows:

Length of the line; this is the *relative* distance from the last picked point (in blue below)

Options for the command in progress (in the gray box)

Angle of line from the x-axis (shown in gray, 31.708 degrees)
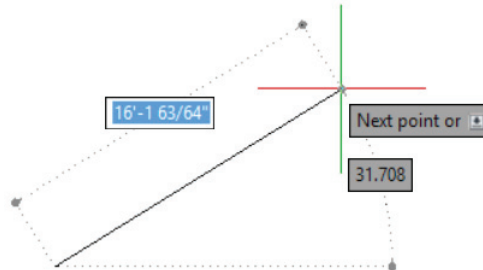
Again, you press **Tab** to switch between the distance and angle fields.

6. To access a command's options, press the keyboard's **Down** arrow. Choose an option.
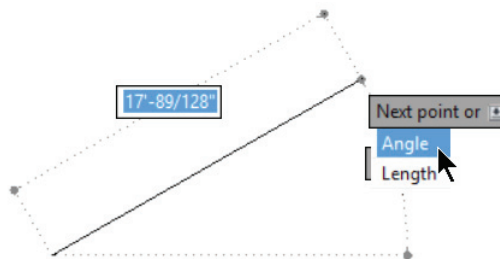
7. Continue drawing.
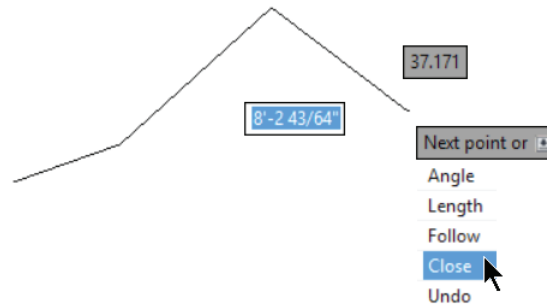
Notice that you can enter options and values at the command prompt or in the dynamic input fields.

There are a number of settings that control the look and function of dynamic input available through the **Settings** command: choose the Dynamic Input tab.

Coordinate Input

Now that you have a familiarity with progeCAD's user interface and how to control the software, the next items to understand are *commands* and *methods of input*.

To draft drawings, you use commands that create and modify entities. Commands also perform utility functions, such as saving and plotting drawings, and extracting information from drawings. You learn how to use progeCAD's commands during the tutorial sections of this book.

Drawings consist of *entities* — lines, circles, arcs, text, dimensions, and so on — that you create and manipulate through inputting commands and coordinates.

Vector Entities

All CAD software works with vector entities. *Vector* means that entities are defined by their geometric features. You never draw the entities themselves; instead, you specify their geometry, and then progeCAD draws the entities.

For instance, instead of drawing a line, you specify the two endpoints of the line:

Command: **line**
Start of line: *(Pick the first end point)*
Angle/Length/<End point>: *(Pick the second end point)*

progeCAD then fills in the line segment between the two points.
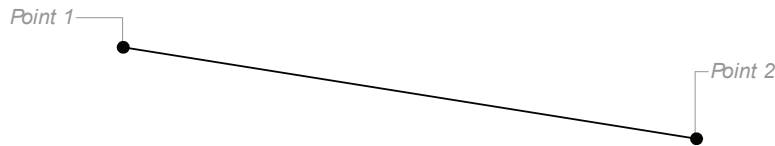
The same technique applies to other entities: progeCAD draws them after you specify their geometric features. Here are a few more examples:

Circles — indicate the location of the center point and the radius; progeCAD draws the circle.

Command : **circle**
2Point/3Point/RADTT/TTT/Arc/Multiple/<Center of circle>: *(Pick the center point.)*
Diameter/<Radius>: *(Specify the radius.)*
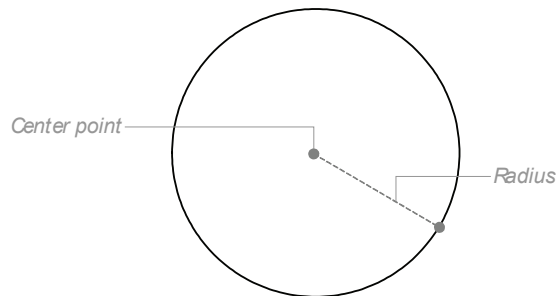
Arcs — indicate two endpoints and another point on the arc.

Command: **arc**
Center/<Start of arc>:
Angle/Center/Direction/End/Radius/<Second point>:
End point:

2: Through point — 1: Start point

3: End point
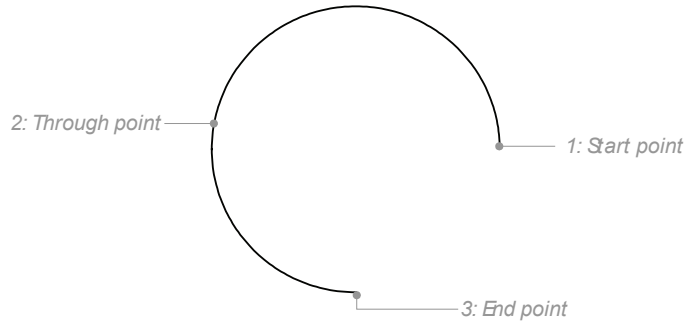
Ellipses — indicate the location and length of two axes.

(Major axis) — 2: Major axis end point

1: Minor axis end point —

(Minor axis)

3: Minor axis end point
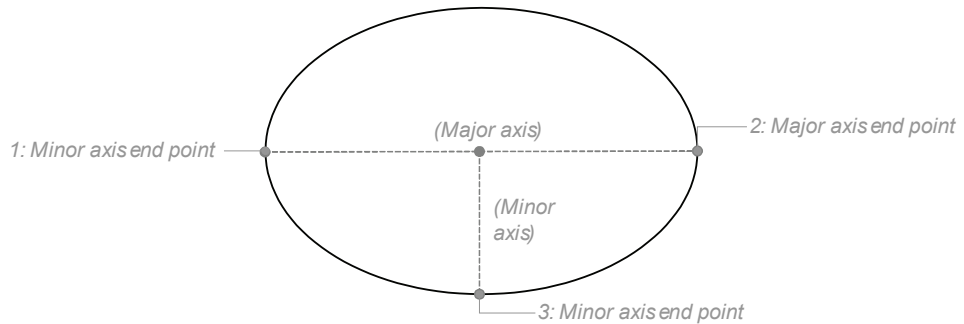
Text — indicate the starting point of the text, and its height and angle.

2: Height — 4: Text

Text

1: Start point —

3: Angle
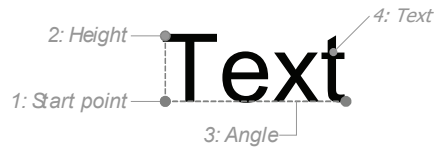
Dimension — indicate the location of two extension lines, and the dimension line and text.

2: Location of the dimension line and text

1.25

1: Start of the first extension line —

2: Start of the second extension line

Wall is 12m long
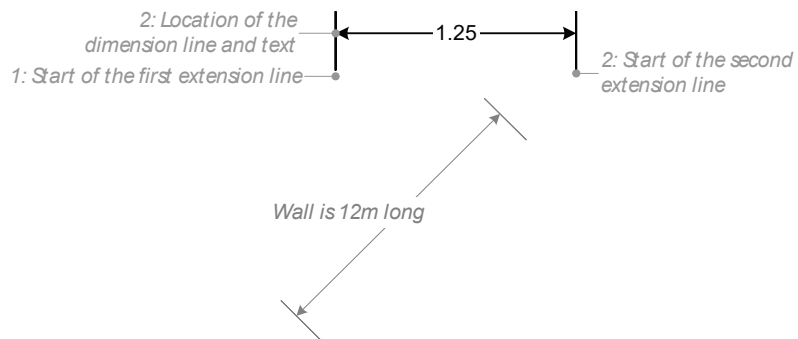
While you can specify line segments by their two endpoints, this may not be the most convenient method. For example, in drawing lines that represent walls, you often know their length, but not where their end points are located.

progeCAD is flexible enough to provide multiple methods of defining entities. To draw walls, for instance, you can specify the start point, its *length*, and the *angle* of the line to the x-axis. Again, you don't draw the line itself, but specify its parameters.
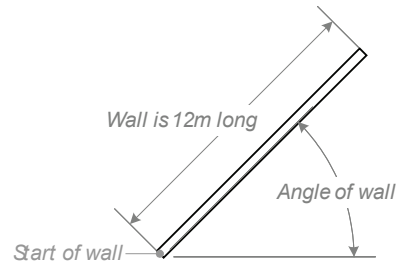
The same is true for nearly all entities created by progeCAD: often, there are two or more ways to draw each of them. The arc, for example, is the most flexible: there are more than a dozen different ways to draw it.

Specifying Coordinates

To draw entities, you need to specify the coordinates (locations) of geometric features. progeCAD provides two methods for specifying geometric features: the keyboard, and the mouse (cursor). The keyboard is used to enter specific numbers, distances, and angles; the mouse is used to show locations in the drawing.

Let's take the **Line** command as an example.

Keyboard Input

The keyboard enters the names of commands and options, as well as to specify exact x, y coordinates, distances, angles, and amounts.

> Command: **line**
> Specify start point, ENTER to use last point or [Follow]: *(Enter coordinate, or enter an option)*

At the "ENTER to use last point/Follow/<Start of line>" prompt you can do any of the following at the keyboard:

- Press **Enter** to continue the line from the last point entered in the drawing.
  > Specify start point, ENTER to use last point or [Follow]: *(Press **Enter**)*

- Press **F** and then **Enter** to specify the Follow option.
  > Specify start point, ENTER to use last point or [Follow]: **f** *(and then press **Enter**)*

- Type x, y coordinates, such as **1.2,3.4**, and then press **Enter**.
  > Specify start point, ENTER to use last point or [Follow]: **1.2,3.4** *(and then press **Enter**)*
  > Next point or [Angle/Length]: *(Enter the next set of coordinates, or enter an option)*

Specifying Coordinates

Although entering x, y coordinates is very common among CAD users, it is not the only way; progeCAD supports a variety methods of inputting coordinates.

x,y — two-dimensional coordinates are used for 2D drafting. A comma (,) separates the x and y values.

Start of line: **1,2**

You can specify coordinates in imperial or metric units.

The z coordinate is fixed by the **Elevation** system variable and is usually set to 0.

x,y,z — three-dimensional coordinates, also known as *Cartesian* coordinates.

Start of line: **1,2,3**

d<a — polar coordinates specify the *distance* and *angle* from the current point. Think of being at the North Pole, and finding any point on the earth by a distance (from the North Pole) and an angle (around the North Pole).

Start of line: **23<45**

The angle bracket (<) prefix indicates the angle. You can specify the angle in degrees, radians, grads, or surveyor's units.

x<a,z — specify cylindrical coordinates, which combine an x-coordinate with an *angle* and a z-*height*. Think of being inside a pop can: the x measures the radius of the can, the z measures the height of the can, and the a measures the angle around the can.

Start of line: 1<45,1

d<a<A — spherical coordinates, the 3D version of polar coordinates. These specify a distance and two angles. Think of being in the center of the Earth: the **d** measures the distance from the center, the **a** measures the angle left-and-right around the equator (x, y plane), while the **A** measures the angle up-and-down along the International Data Line (y, z plane).

Start of line: **1<45<90**

Remember that commas (,) separate distances, while brackets (<) signify angles. There are additional symbols that modify coordinates:

Negative distances and angles — dash (-). Negative angles are measured clockwise from the x axis, and are indicated by the dash prefix.

Start of line: **-1,-2<-45**

(Positive angles are measured counterclockwise from the x axis, although the direction can be reversed with the **Units** command, as can the location from which they are measured.)

Relative coordinates — at (@). Most coordinates are relative to the origin of the drawing (0,0) and are called "absolute coordinates." By placing the @ symbol in front of the coordinate, you measure distances and angles relative to the last point picked in the drawing.

Start of line: **@1<45**

End point: **@7,5**

UCS in WCS coordinates — asterisk (*). When used in a user-defined coordinate system, the * prefix forces coordinates to be interpreted relative to the world coordinate system.

Start of line: ***1,2,3**

Point Filters

Finally, you can enter partial coordinates by using *point filters*. "Point" refers to the dot (.) that you first enter to alert progeCAD that you are entering a partial coordinate. "Filter" means that the coordinates are being provided partially: first x, y, z, or some combination thereof, and then the "missing" coordinate(s). The table below lists all possible combination of point filters:

Point Filter	Specify...	...and then specify
.x	Select x	Still need y, z
.y	Select y	Still need x,z
.z	Select z	Still need x,y
.xy	Select x,y	Still need z
.xz	Select x,z	Still need y
.yz	Select x,x	Still need x

Well, not finally. Coordinate input can also be modified by entity snaps, as described later in this chapter.

Direct Distance Entry

progeCAD has a "hidden" form of coordinate entry known as "direct distance entry." Here you move the cursor in a specific direction, and then type in the distance. This is an interactive form of relative coordinates (@d<a).

Here's how to use direct distance entry with the **Line** command:

Command: **line**
Start of line: **1,2**
Next point or [Angle/Length]: *(Move mouse/cursor in any direction, and then enter a distance:)* **5**

While you can move the cursor in any direction to indicate the angle, it's more accurate to use ortho mode or an esnap mode to assist you in specifying the angle.

Units of Measurement

progeCAD accepts distance input in a variety of formats, including decimal (metric) and feet-inches. Angles can be input in degrees, radians, grad, and surveyor's units. Internally, however, progeCAD converts them to unitless scientific notation.
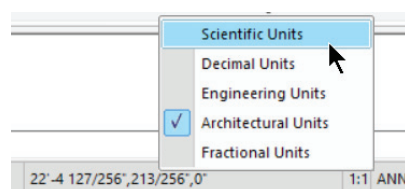
Note that you can enter distances and angles in any units format, even when it is different from that displayed by the status bar. You can change the display of units at any time by right-clicking the coordinate display on the status line.

Mouse Input

You most commonly use the cursor to show progeCAD locations in drawings; the mouse (or touch screen digitizer) controls the cursor. The problem with using the mouse is that it is not particularly accurate by itself. You would find it difficult to draw, for example, a 31.01"-diameter circle using just the mouse.

For this reason, progeCAD provides several aids for more accurately drawing with the mouse and cursor.

Status Bar shows the distance (**Length:**) and angle (**@ Angle:**) from the last pick point. As you draw, keep an eye on the status bar to see the size of the entity. (See figure below.) The drawback is that watching the status bar is too slow for production use.
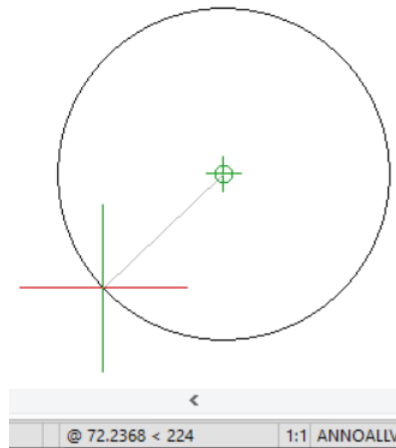
Snap specifies the cursor resolution. For instance, when you set the snap distance to 1", then mouse movements in the drawing area are accurate to the nearest one inch. (The snap does not affect the cursor outside of the drawing area.) Use the **Snap** command to specify the snap distance.

The snap distance is invisible, however, so many users also turn on the grid to match the snap distance. (In the figure below, the grid is visible as an array of lines.)
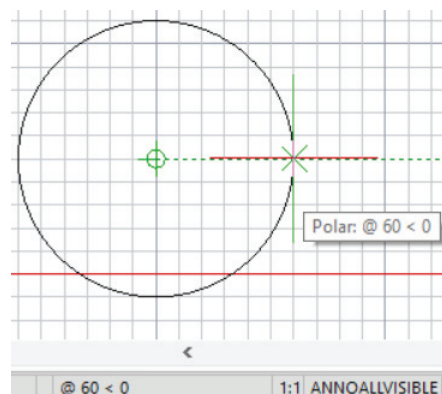
Entity Snap causes the cursor to snap to geometric features of entities, such as their end, mid, and tangent points. This is useful when you want to draw from one geometric feature to another, such as drawing a line from the precise center of one circle to the precise center of another, as illustrated below.
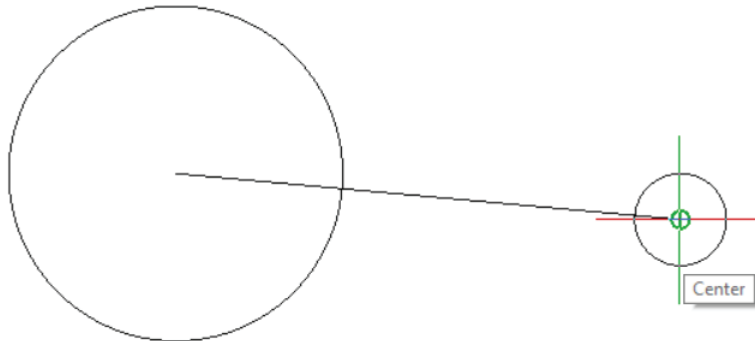
To set entity snap modes, enter the **OSnap** command. (Entity snaps are called "esnaps" for short, but are also known as object snaps, or "osnaps.") To further help you, progeCAD displays markers and tooltips that identify geometric features. All of progeCAD's esnap features are listed later in this chapter.

Entity Tracking lets the cursor find geometric features on nearby entities, such as a corner or a line.

Polar Mode lets the cursor find increments of commonly-used angles easily, such as 15 or 45 degrees.
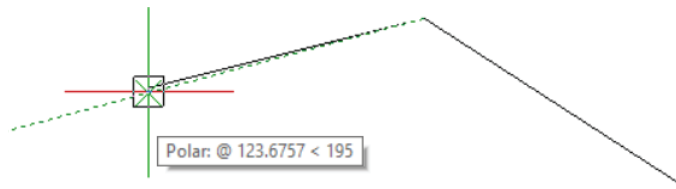
Selecting Entities

When it comes to editing drawings, you need a way to select the entities to be edited. *Selection* tells progeCAD which group of entities you need to modify. In most cases, you must use the mouse to select entities; in a few cases, you use the keyboard instead.

There are two ways to select entities:

Before starting an editing command; this is called "grips editing"

After starting the editing command; this is called "entity selection"

progeCAD lets you use whichever method you find easier!

Grips Editing

Here's how to perform grips editing: With no command active, select one or more entities by picking them with the cursor. (To select all entities in a drawing, press **Ctrl+A** on the keyboard.)

Notice that the entities change to dashed lines and that blue squares appear; this is how progeCAD responds, letting you know which entity you picked. The blue squares are called "cold grips."
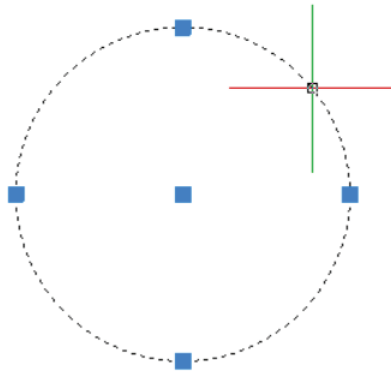
Move the cursor over a grip, notice that it turns orange. This is how progeCAD lets you know that the grip is ready to be picked.
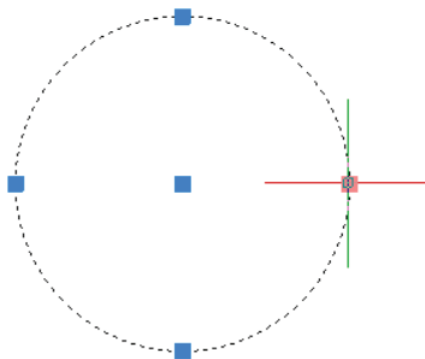
Click on the orange grip. Notice that it turns red. This is called a "hot grip." The color tells you that you can now move the grip with the cursor. Dragging a grip relocates it and changes (edits) the entity.

In the figure below, I am making the circle larger.
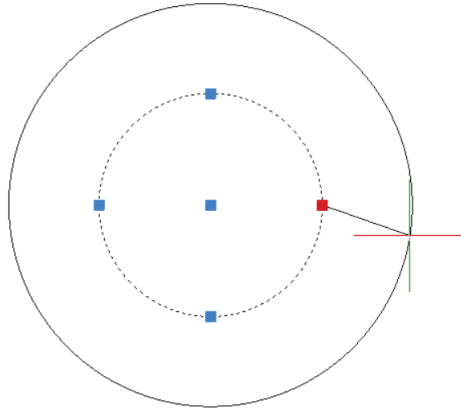
Depending on the entity and which grip you move, you either change the entity's size (stretch) or move the entity's position. Check the command line for a description of the grip action. Examples include:

> New end of line: *(Changes length and angle of the line.)*
> New location for line: *(Moves the line.)*
> New circle radius: *(Changes the size of the circle.)*
> New basepoint for text: *(Moves the text.)*

To erase selected entities, press **Del** on the keyboard.

To cancel grips editing, press the **Esc** key.

To see a list of additional editing commands, press the mouse's right button. The resulting short-cut menu is illustrated below, and lists the names of dozens of editing commands that operate on selected entities. For example, selecting **Delete** from the menu erases the selected entities.
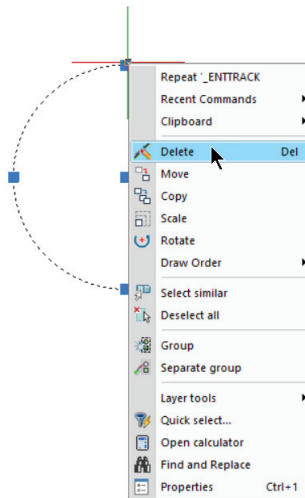
To change the size and color of grips, use the **DdGrips** command.

You select a single entity by picking it with the cursor. Ensure part of the entity is within the cursor's pickbox ⬚. To select additional entities, just keep right on picking them.

After a while, picking entities one-by-one becomes tedious, and so progeCAD provides ways to pick groups of entities. One method is the *selection rectangle*. It selects entities within an area specified by a rectangle, which is created like this:

1. Pick a blank point in the drawing (away from any entities).
2. Form a rectangle by moving the cursor to the left or to the right.

    • **Moving the cursor right** forms a *windowed* selection. progeCAD selects all entities that fall entirely within the selection rectangle. Entities crossing the rectangle and outside the rectangle are ignored.

    Notice that progeCAD displays an icon near the cursor to remind you of the type of selection rectangle you are creating. For windowed selections, the cursor shows a circle entirely within a rectangle and the color blue.

    • **Moving the cursor to the left** forms a *crossing* selection. progeCAD selects all entities within and crossing the selection rectangle. For crossing selections, the cursor shows a circle crossing the rectangle and the color green.

Selection Set Modes

But there are more selection modes! Selection areas need not be rectangles; they can also be circles, polygons, and fences. Typically, editing commands asks you to select objects, as follows:

```
Command:  copy
Select entities to copy: cp
```

At the "Select entities" prompt, you can pick entities one at a time with the mouse, or enter one of the following options. You enter the highlighted letters only. If prompt box is displayed, you can choose a selection option from it.
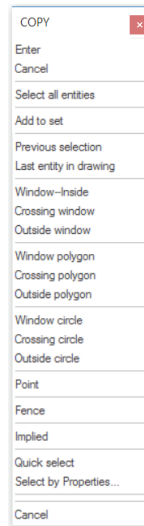
Select	Comments
Add to set	Adds one or more entities to the selection set (can also press **+**).
Crossing **c**ircle	Selects entities contained within or crossing the boundary of a circular selection area.
Crossing **p**olygon	Selects entities contained within or crossing the boundary of a polygon selection area.
Crossing window	Selects entities contained within or crossing the boundary of a rectangular selection area.
Fence	Selects entities crossing a line or line segments.
Last entity in drawing	Selects the entity most recently added to the drawing.
Outside **c**ircle	Selects entities falling completely outside a circular selection area.
Outside **p**olygon	Selects entities falling completely outside a polygon selection area.
Outside **w**indow	Selects entities falling completely outside a rectangular selection window.
Previous selection	Selects entities included in the previous selection set.
Select **all** entities	Selects all entities in the current drawing (**Ctrl+A**).
Subtract from set	Removes one or more entities from the selection set (**-**).
Window inside	Selects entities contained entirely within a rectangular selection window.
Window **c**ircle	Selects entities contained entirely within a circular selection window.
Window **p**olygon	Selects entities contained entirely within a polygon selection window.

The list above contains selection modes that are best for selecting entities based their location in the drawing.

A second method is to select them by their *properties*. Notice the **Select by Properties** option in the prompt box above. When you choose it, a second prompt box appears:
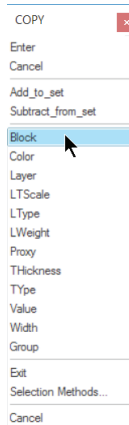
Select	Comments
Block	Selects all blocks of a specified name
Color	Selects all entities of the specified color
Layer	Selects all entities residing on the specified layer
LTScale	Selects all entities with a specified linetype scale factor
LType	Selects all entities of the specified linetype
LWeight	Selects all entities of the specified line weight
Proxy	Selects all proxy entities
THickness	Selects all entities of the specified thickness
TYpe	Selects all entities of the specified type
Value	Selects all text with the specified words
Width	Selects all polyline entities of the specified width
Group	Selects all groups of the specified name

Add to set and **Subtract from set** allow you to add more entities based on their properties.

The **Exit** option returns you to the previous cursor menu. **Selection Methods** displays the Drawing Settings dialog box. **Cancel** cancels the editing command in progress.

TIP   You can mix keyboard and mouse entries. For instance, the prompts below show a line drawn using point filters, esnap modes, mouse/cursor picks, and relative coordinates:
    Command: **line**
    Start of line: **.xy**
    Select x,y of: **mid**
    of *(Pick an entity with the mouse/cursor.)*
    Still need z: **3**
    Next point or [Angle/Length]: **@1<-45**

Quick Select

I'm not sure why it's called Quick Select, but there is one more way to select entities. This is using the **QSelect** command and its dialog box.
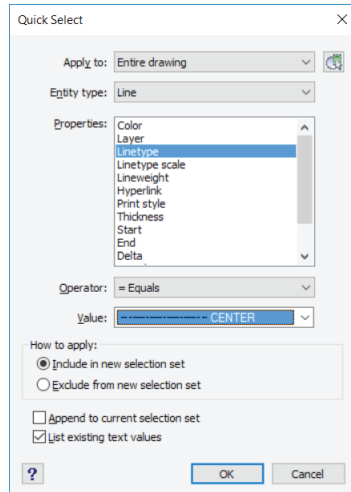
It lets you select entities with a specific property, such as all lines with the Center linetype.

141 Tips

the 141 tips listed in this chapter were at one time included in a *icad.tip* file that accompanied older releases of the program. I include them here, because they contain userful information for working more productively with progeCAD. Some tips are updated for 2017 or edited for clarity.

User Interface

Tip #1. You can right-click different areas on the screen to bring up different cursor menus. Try doing this on the status bar at the bottom of the progeCAD window.

Command Help

Tip #2. You can press **F1** at any time to display progeCAD Help. When you do this during a command, progeCAD displays help for that command specifically. progeCAD provides the following:

- Online user's guide
- Online Help, which gives details about commands, tools, aliases, and system variables
- Online developer references for programmers

Tip #3. You can learn about any command listed on a menu by highlighting the item, and then reading the text in the status bar at the bottom of the main window.

Screen

Tip #4. You can change the color of your screen background: from application menu or the **Tools** menu, choose **Options**, and then choose the **Display** tab. Click the **Color Scheme** button, and then choose the background color you want.

Tip #5. You can maximize the drawing area on the screen by hiding items you do not need at the moment: toolbars, command bar, scroll bars, and status bar. To hide or show elements of the user interface, select or clear the item that you want to hide or show in the **View** menu's **Display** submenu (CAD Classic workspace).
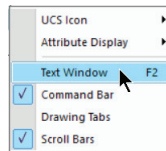
Tip #6. progeCAD can display a reference grid and snap to regularly spaced points on the screen. To see these options, enter the **Settings** command, choose the **Coordinate Input** tab, and then from the **Change Settings for** droplist, select **Snap and Grid**.
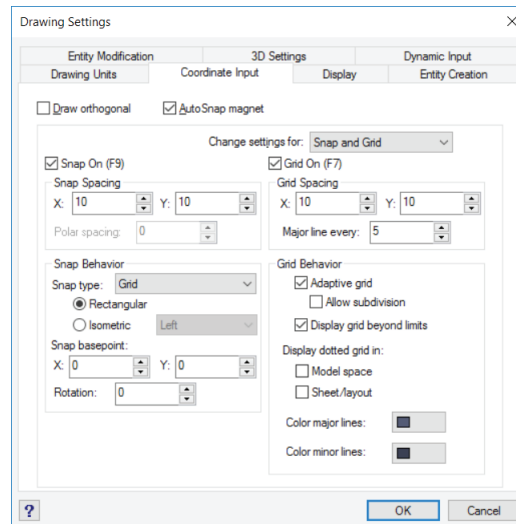
Tip #7. The drawing crosshairs use a different color for each axis. To select the colors you want, enter the **Options** command, and then click the **Crosshairs** tab.
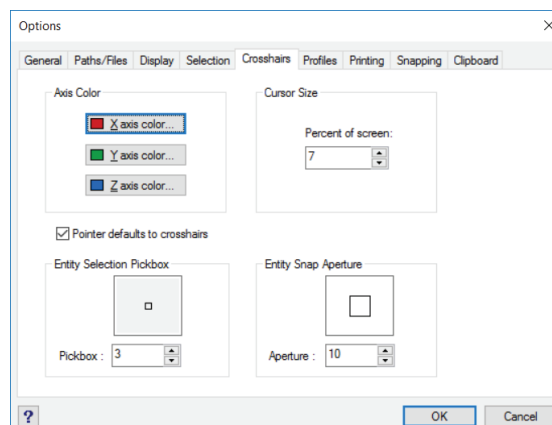
Tip #8. You can show a cursor menu of commands to modify specific entities. Select an entity, and then right-click to see the cursor menu.
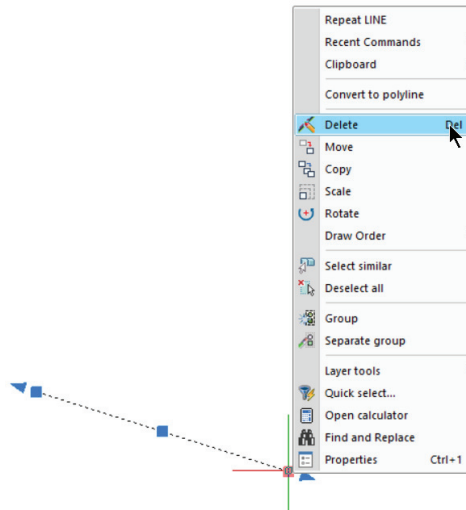
Command Bar

Tip #9. If you are accustomed to using the command line in other CAD packages, such as AutoCAD, you can type the same commands and use the same options in progeCAD — even when you do not see them on the screen. Simply type the commands, and then watch the status bar. When the command bar is not displayed, you can see each step of an active command in the status bar.

Specify start point, ENTER to use last point or [Follow]:

Tip #10. If you prefer to hide the command bar for more screen space, double-click the status bar.

Tip #11. You can specify any word(s) or symbol for the command prompt, such as 'Ready' or 'OK'. To change the text appears in the command bar, enter **CmdLnText**, and then enter the new word or symbol.

Tip #12. You can dock the command bar at the top or bottom of the progeCAD window or float it anywhere you want.

Tip #13. You can change the number of lines displayed by the docked command bar by dragging the separator line.
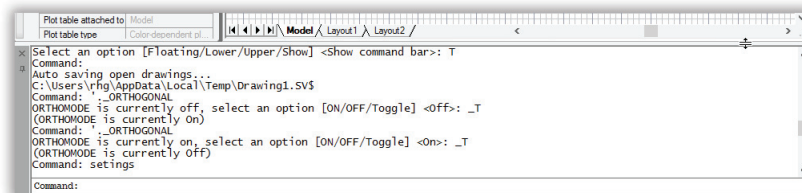
You can resize the floating command bar: drag it to the middle of the window, change the size, and position it where you want. To prevent the command bar from docking, press and hold the **Ctrl** key as you move it.

Tip #14. To see all recent commands, prompts, and input, press **F2** to see the Prompt History window.

Tip #15. You can display entity information in the Prompt History window by typing **List**. Follow the prompts to choose how you want information displayed. Alternatively, use the Properties palette.

Tip #16. progeCAD automatically tracks your command history. The default number of recorded lines is 256, but you can change the number to between 1 and 400. Enter the **Options** command, and then click the **Display** tab. Enter the number of lines you want to track next to **Command lines to track**.

Tip #17. A number of commands — such as Zoom and Pan — can be *nested*: this is when a second command is started while another is still in progress. Type the apostrophe ('), and then enter the second command. While the nested command is active, an angle bracket (>) is displayed in the command prompt.

Prompt Box

Tip #18. After you select a command, an optional prompt box displays the options of the command. You use the mouse to select command options from the prompt box, such as for the PLine command shown below.
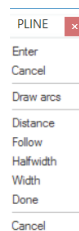
Tip #19. You can turn the prompt boxes on and off. Enter the **Options** command, click the **Display** tab, and then change the check box fo**r Display Prompt Boxes**.

Tip #20. You specify command options by typing the capital letter(s) shown in the command bar. For example, to choose the **Length** option during the **Line** command, simply type **L** at the prompt.

Tip #21. To change where prompt boxes appear on the screen, move one to a different location. Future prompt boxes will display in the new position.

Menus

Tip #22. You can set progeCAD to show the tools and menu items appropriate for your experience level. Enter the **Options** command, choose the **General** tab, and select **Beginner**, **Intermediate**, or **Advanced** from the **Experience Level** droplist.

Tip #23. On menus, a check mark indicates that the setting is in use.

Tip #24. You can save your arrangements of menus (by exporting them) and you can read menus from AutoCAD and other CAD programs (by importing *.mnu* and *.cui* files). To export and import menus, enter the **Customize** command, choose the **Menus** tab, and then click the **Import** or **Export** buttons.

Tip #25. You can replace all your menus with a new set, but it is better to append a new ones to the existing menus. Click the **Append To Current Menu** check box.

Dialog Boxes

Tip #26. You close dialog boxes by pressing the **Esc** key on the keyboard.

Toolbars

Tip #27. You can move the toolbars and the command bar to any place on the screen by dragging them away from the edge of the window.

Tip #28. You can make the tool icons larger to help you recognize them more easily on high-resolution screens. To change to large toolbars, right-click any toolbar, choose **Toolbars**, and then select the **Large Buttons** check box.

Tip #29. You can remove from toolbars the buttons you rarely use. Hold down the **Shift** key as you drag the tool away from the toolbar. For more toolbar customization options or to replace tools you have removed, enter the **Customize** command.

Tip #30. Several toolbars include *flyout* toolbars with related commands and options. Flyout toolbars are indicated by a small, black triangle in the lower right corner of the tool.

Tip #31. You can display as many or as few of the toolbars as you want. Right-click any toolbar, choose **Toolbars**, and then click the check box for the toolbars that you want to display.
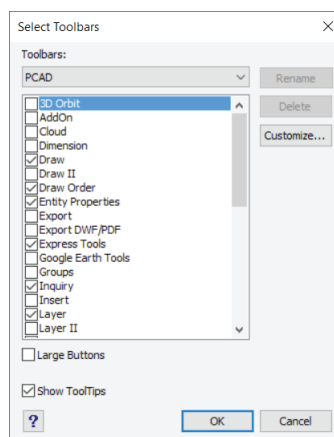
Tip #32. To hide a toolbar, drag it away from the edge of the window, and then click the **X** button.

Tip #33. You can display a cursor (shortcut) menu of available toolbars by right-clicking any toolbar. In the shortcut menu, choose **PCAD**.

Status Bar

Tip #34. You can change the units display method by right-clicking the coordinates display in the status bar. Choose a different set of units from the cursor menu.

Dynamic Pan and Zoom

Tip #35. progeCAD provides dynamic pan and zoom using the mouse:

- To pan dynamically, hold down the middle mouse button (or roller wheel) and then move the mouse about
- To zoom dynamically, scroll the mouse's roller wheel back and forth

Tip #36. You can pan horizontally and vertically using the scroll bars at the side and bottom of the drawing window. The length of each scroll bar is proportional to the view size.

Drafting and Drawing

Creating Drawings

Tip #37. You can use any drawing you want as the basis for future drawings; these are called "templates." Template include borders, title blocks, and your preferred settings.

Tip #38. To use the New Drawing wizard, go to the application menu (or **File** menu) and then select **New**. Or else enter the **PNew** command. Click the **Use a Wizard** button.
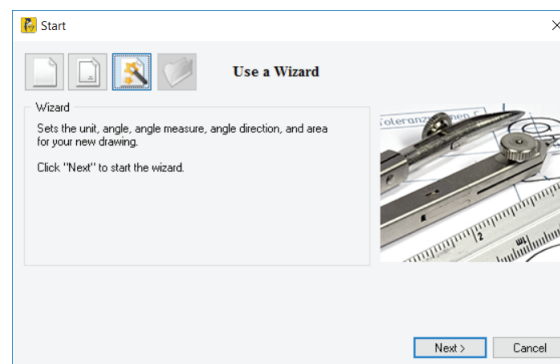
To open a new drawing *without* the assistance of the wizard, enter the **New** command.

Tip #39. You can open multiple windows to view your drawing in different ways at the same time. Right-click a drawing tab, choose **New**, and then open the same drawing.

Tip #40. You can include progeCAD drawings in the documents of other programs. Simply drag and drop the progeCAD drawing onto the document. To edit the drawing in-place, double-click it.

Tip #41. Similarly, you can drag and drop documents from other programs into progeCAD drawings. For example, you could add a Libre document or a Visio diagram.

Information from the Drawing

Tip #42. You can find distances, areas, and exact locations in your drawings using the tools on the **Inquiry** toolbar.

Tip #43. You create drawings full-size in model space (the default), and then arrange it any way you want for printing in paper space.

Blocks

Tip #44. You can create *blocks*, which are combinations of entities that you use repeatedly. To create a block, draw something you want to reuse. To do so, enter the **Block** command and then choose the objects you want to make part of the block.

To use the block later, use the **Insert** command, or else select the block in the progeCAD Explorer.

Tip #45. progeCAD Explorer lets you view entire collections of blocks before choosing one to insert in the drawing.
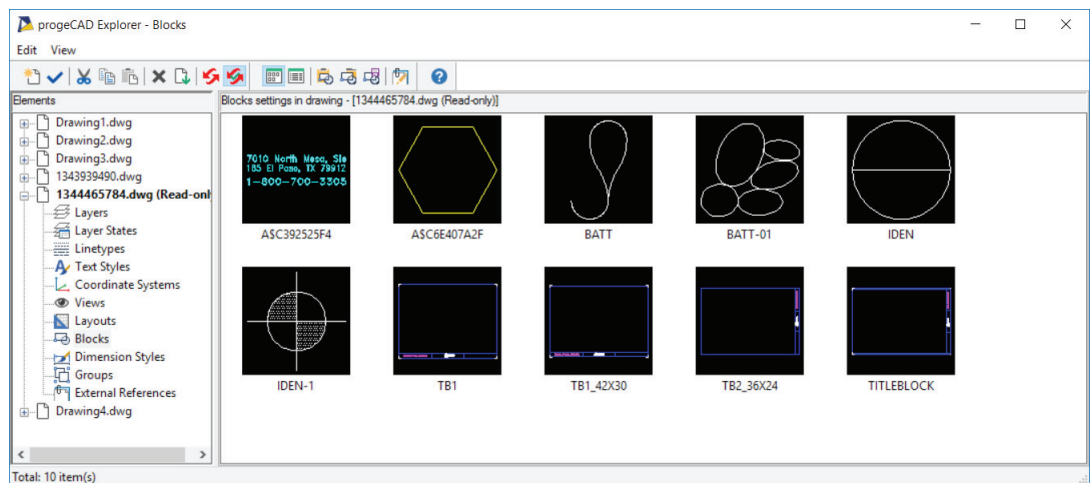
Tip #46. You can associate text (called "attributes") with blocks as you define them. Each time you insert the block, you enter the appropriate information for that instance, such as a part number, price, or supplier.

Tip #47. The array of blocks inserted with the **MInsert** command is itself a block.

Tip #48. You can insert blocks (and hatches) in exploded form so that they are placed as pieces rather than as a single unit. To do this, precede the name of the block (or hatch pattern) with an asterisk (*), or else check the **Explode** options in the Insert dialog box.

Tip #49. You can use block attribute information in an external drawing database. To export data from the attributes in your drawings, use the **EAttExt** command.

Tip #50. You cannot explode blocks inserted with the **MInsert** or **Xref** commands.

progeCAD Explorer

Tip #51. You can see and work with the layers, linetypes, styles, and other elements of external references (xrefs) in the progeCAD Explorer. Open the explorer with the **Explorer** command.
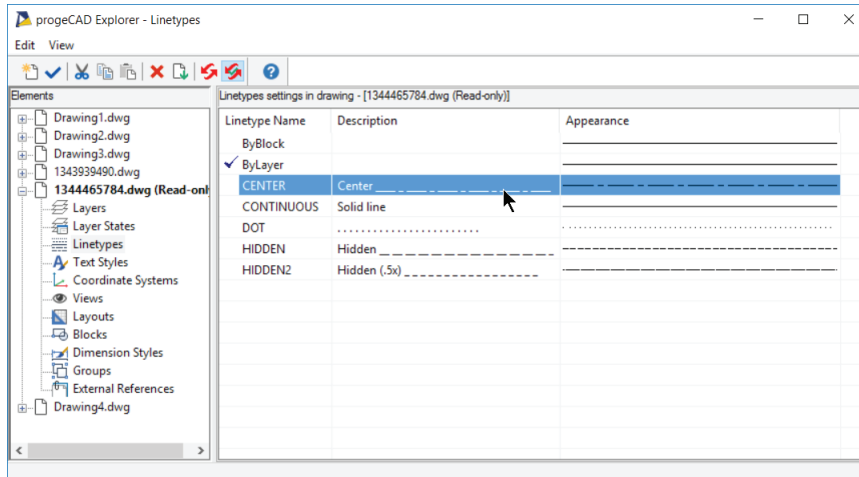
Tip #52. You can save a block to a separate drawing file using the progeCAD Explorer. Select the block, and then click the **Save Block** tool to save it as its own *.dwg* file.

Tip #53. In progeCAD Explorer, you can double-click the name of an item to make it current. For example, to set a layer as the current layer, double-click the name of the layer.

Tip #54. You can copy anything in the progeCAD Explorer from one drawing to another: layers, linetypes, saved views, blocks, text styles, dimension styles, user coordinate systems, and more.

Tip #55. You can use the progeCAD Explorer to purge unused layers, linetypes, styles, coordinate systems, views, blocks, and dimension styles from drawings. Right-click the item you want to remove, and then choose **Purge**.

Tip #56. To make information from external references available in the progeCAD Explorer, reload the drawing using the External Reference (**XRef**) command.

Tip #57. You can rename elements from within the progeCAD Explorer.

Other Drafting Tips

Tip #58. You can constrain lines and other entities to draw only vertically and horizontally. For just a few lines, hold the **Shift** key down while you draw. To work this way continually, click the **ORTHO** button on the status bar.

Tip #59. You can place point entities or blocks at regular intervals along an entity using the **Divide** and **Measure** tools:

- *Divide* marks the entity in a specified number of segments
- *Measure* marks off segments of specified length

Neither command alters the original entity.

Tip #60. You can draw quickly and precisely by snapping to points on existing entities, such as the end of a line or center of a circle. To set entity snaps and see their current states, use the **Entity Snaps** toolbar.

Tip #61. *Polylines* are sets of line and arc segments handled as a single entity. Polylines can have qualities that ordinary lines cannot, such as width. When you move a vertex of a polyline, both adjacent segments move with it.

Tip #62. When drawing lines, arcs, and polylines, you can use the **Follow** option to continue drawing from the last point and in the same direction.

Tip #63. A *donut* is a planar, circular entity that has an inside and outside diameter, like a flat washer. (Planar means the object lies in a plane, such as the x,y-plane, and so it is not a 3D object.) You can draw donuts by using either the two-point, three-point, or tangent-radius method.
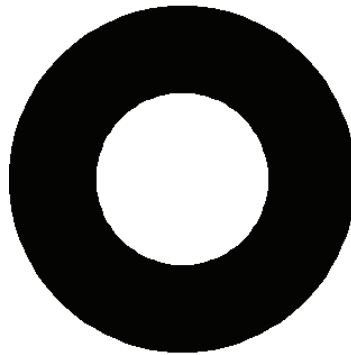
Tip #64. When you create planar entities using the **Solid** command (a.k.a Plane), you can constrain the planes to squares, rectangles, and equilateral triangles.

Tip #65. You use the **Rectangle** command to draw squares and rectangles at a specific angle. You can also adjust the width of the lines that make up the shape.

Tip #66. Your drawing may contain blocks that you no longer need. Use the **Purge** command to clean out unused blocks.

Tip #67. You can use the **Center** entity snap to snap to the center of any two-dimensional closed polyline, including rectangles and polygons, as well as to the centers of circles and arcs.

Tip #68. You can bisect lines, arcs, and polylines using the Ray command. Start the **Ray** command, select **Bisect**, and then select **Entity**.

Tip #69. You can create regular polygons such as hexagons and pentagons using the **Polygon** command. All polygons are created as closed polylines.

Tip #70. Polylines have a direction. To reverse the direction of a polyline, first select it, and then enter the **PEdit** command; choose then **Reverse** option.

Tip #71. You can esnap lines perpendicular (or tangent) to existing entities. Simply select a point on the entity with **PERpendicular** (or **TANgent**) entity snap turned on, and progeCAD adjusts the point so that the line you are drawing is perpendicular (or tangent) to the entity. (Tangents are made to circular objects only, like circles and arcs.)

Tip #72. A drawing can contain a reference to another drawing, called an "external reference," or *xref*. The display of the referenced drawing updates each time the reference is reloaded using the **XRef** command.

Tip #73. When drawing lines, you can specify the angle (or length) first using the **Angle** (or **Length**) option, and then adjust the other parameter dynamically.

Tip #74. You use the **PEdit** command to specify whether linetypes are broken or continuous across vertices. Select the entity, and then enter the **PEdit** command. Choose the **Linetype-mode** option, and then enter On or Off.

Tip #75. You can join collinear lines and coplanar arcs with the same center and radius using the **Join** command.

Tip #76. If you are drawing in isometric planes and the grid is on, the x axis grid spacing is automatically adjusted to isometrically match the y axis spacing.

Tip #77. To draw isometrically, enter the **Settings** command, choose the **Coordinate Input** tab, and in the **Change Settings For** list, click **Isometric Snap and Grid**.

Tip #78. When using the **Polygon** command, you can change the line width of the polygon to be drawn.

Tip #79. To close an arc, use the **Arc** option of the **Circle** command.

Tip #80. You can set point entities to be displayed in one of twenty ways. Enter the **DDPtype** command, and then choose one of the display options.
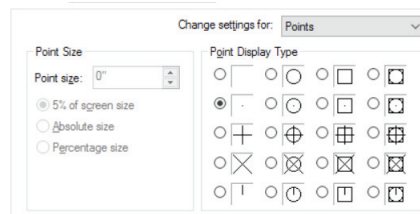
Tip #81. You can make a layer current by selecting an entity on that layer. Select an entity on the layer you want to make current, and then enter the **LayByEnt** command.

Tip #82. The length of the last entity you specified or drew before starting the **Lengthen** (aka EditLen) command is displayed by the prompt.

Tip #83. To create a planar (2D) face with the **Face** command, position all points on the same plane.

Tip #84. When **Fill** is turned off, the program displays and plots filled drawing entities more quickly on slower computers.

Tip #85. For more accurate tracking during the **Sketch** (a.k.a. Freehand) command, specify shorter segments. Doing this results in a smoother freehand entity, but creates larger drawing sizes.

Tip #86. Displaying objects as icons with the **InsertObj** command is faster than displaying the inserted OLE objects themselves.

Tip #87. Draw isometric circles with the **Isocircle** option of the **Ellipse** command.

Tip #88. To create a polygon mesh with the **RuleSurf** command, select each entity at the same ends. To create a self-intersecting polygon mesh, select each entity at opposite ends.

Tip #89. Entities that you preselect with the **Select** command are placed into a *previous* selection set. When you perform many progeCAD commands (such as Delete or Change), you can choose the Previous selection set by entering 'p' at the 'Select entities' prompt.

Tip #90. The current layer name is displayed in the Properties toolbar.

Tip #91. To continue text under the previous line, start the **Text** command and then press **Enter** until it displays the 'Text:' prompt.

Tip #92. progeCAD provides three visual indicators to show that an entity snap is turned on:

- Status bar — when any entity snap is turned on, and esnap icon appears blue and/or **ESNAP** appears in black
- Object Snaps toolbar — entity snaps that are on are displayed with a blue outline

- When drawing or editing — the cursor displays icons indicating the current entity snap

Tip #93. The **Flatten** command allows you to move sets of entities, such as three-dimensional walls, to a new elevation. When you are in plan view, this command has no visible effect.

Tip #94. When the grid spacing is **0**, it is not turned off. Rather, the grid is set to the same spacing as the snap spacing.

Tip #95. Use the **Style** option of the **Snap** command to turn on isometric drawing mode. In addition, turn on the grid to help you visualize the isometric grid.

Tip #96. When the system variable **LimCheck** is turned on, you are restricted from drawing entities outside the rectangular area you define when you use the **Limits** command:

- To draw entities outside the drawing limits, set the **LimCheck** variable to **0**
- To restrict drawing to within the drawing limits, set the **LimCheck** variable to **1**

Tip #97. The **Line** command's **Follow** option uses the angle stored in the **LastAngle** system variable.

Tip #98. Use the **Purge** command to purge unused elements, such as empty layers, unused scales, and unused linetypes. Unused colors and lineweights cannot be removed.

Tip #99. There are two ways to chamfer entities: you can specify the distances from the intersection point of each entity, or you can specify the distance on the first entity and an angle for the chamfer line.

The distance you specify for the **Chamfer** command's first chamfer distance is saved in the **ChamferA** system variable. The distance you specify for the second chamfer distance is saved in the **ChamferB** system variable.

To draw rectangles with the **Rectangle** command with chamfered corners, set the chamfer distances with the Chamfer option.
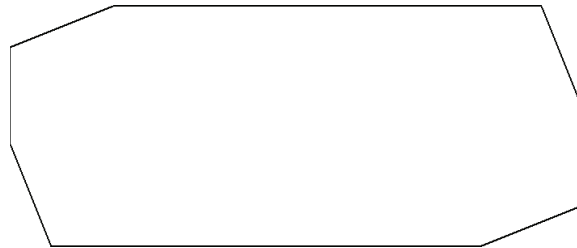
Tip #100. The distance you specify for the **Rectangle** command's fillet distance is used for all subsequent rectangles you draw. To change the fillet distance, choose **Fillet** again and enter a new value.

Tip #101. progeCAD supports the standard Windows ANSI character set. To enter an extended character, press **Alt** plus the numeric equivalent (be sure that **Num Lock** is turned on) on the keyboard's numeric keypad. For a list of the ANSI characters supported and their numeric equivalents, refer to the Character Map utility included with Windows.

Tip #102. You cannot use the **View** command's **Save** and **Restore** options as a transparent command when you are in paper space, during the **Viewpoint** or **DView** commands, or when zooming or panning the display.

Editing

Tip #103. You can apply the properties of one entity — including color, linetype, and thickness — directly to other entities. Enter the **MatchProp** command.

Tip #104. If you press **Enter** to select all entities as boundaries during the **Trim** command, the program immediately continues to the next prompt.

Tip #105. You can simultaneously modify the properties of any number of selected entities. Enter the **Properties** command, select the entities, and then make the changes that you want.

Tip #106. You cannot edit the length of a polyline that has been splined or fitted.

Tip #107. To edit the attributes of more than one block at a time, use the **BAattMan** command.

Tip #108. You use the **Explode** command to alter various properties of entities as they are broken into their component pieces.

Tip #109. You use the **Copy** and **Move** commands between drawings. When prompted for a displacement point, specify a point in the destination drawing.

Tip #110. You can fillet or chamfer selected vertices of a polyline. Select two points on the polyline, and all vertices between the selected points will be modified. The operation goes in the direction of the polyline, so on closed polylines the order of point selection determines which vertices are modified.

Tip #111. You can use the system variable **MirrText** to specify whether you want text mirrored during the **Mirror** command:

- To change the text direction, set the **MirrText** variable to **1**
- To retain the text direction, set the **MirrText** variable to **0**

Tip #112. The **Fillet** command fillets closed polylines by two different methods, depending on how the polylines were closed:

- If you used the **Close** option of the **Polyline** command, then all vertices are filleted
- If you specified the endpoint of the last segment, the last vertex is not filleted, even if you used entity snap to place the endpoint

Tip #113. You can change entities to make them the same length. Enter the **EditLen** command, and then choose the **Total** option. Enter the length you want, and then select the entities you want to lengthen.

Tip #114. You can alter the size of entities directly by moving their grips. Select an entity, and then click on a grip. In general, grips at the center move the entity, while those on the periphery change its size.

Tip #115. You can fillet a polyline, even if the intersection lies on a curved segment of the polyline.

Tip #116. You can change the hatching style for nested regions to hatch either the outer and inner regions, only the outer region, or all regions. Click the **More Options** button in the Hatch command's dialog box, and then change the Island Detection Option section.
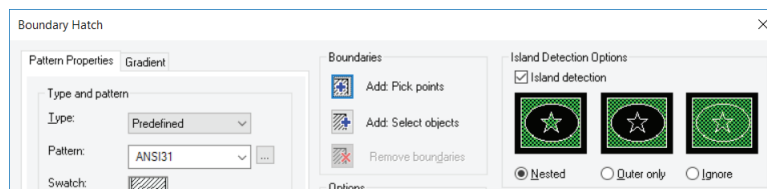
Tip #117. You can erase the ends of sketches you are drawing using **Sketch**. In the prompt box, select **Delete On** and move the cursor to erase. To continue drawing, select **Delete Off**.

Tip #118. You can straighten polyline segments. Select the entity, and then right-click the grip at a vertex. Choose the **Remove Vertex** option from the shortcut menu.
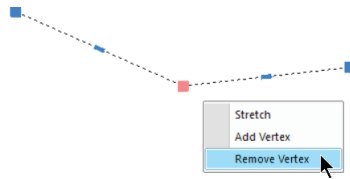
Tip #119. You can also change the individual polyline segments between lines and arcs. Select the entity, and then right-click the grip at the center of a segment. Choose the **Convert to Arc** option (or Convert to Line) from the shortcut menu.

Tip #120. You can explode traces, planes, and 3D faces into their component lines. Enter the **Explode** command, and then select the entities to explode.

Tip #121. When editing a polyline, you can taper the width of the polyline continuously from start to end. All vertex widths are automatically calculated. Pick the polyline, and then enter the **PEdit** command: choose the **Taper** option.

Tip #122. You can convert circles into arcs with the **Break** command: pick two points between which you want a portion of the circle removed.

Selection Sets

Tip #123. You can create a selection set and add entities to it by their properties, such as layer or color. Enter the **Select** command, and then choose the **Filter** option.

Tip #124. To do the same thing as tip#123 with a dialog box, enter the **QSelect** command.

Tip #125. You can adjust the size of the cursor used for selecting entities. To set the size, enter the **Settings** command, and then choose the **Coordinate Input** tab. In the **Change Settings For** list, choose **Entity Selection**.

Tip #126. You can select several entities and change their properties simultaneously. Select the entities, and then right-click to see the cursor menu. Select **Properties**, and then make the global changes.

UCS and WCS

Tip #127. The World Coordinate System (WCS) is the default and the foundation on which user coordinate systems are built. You can define your own coordinate systems to make specifying points and reading entity information easier: create a new UCS in the progeCAD Explorer.

You return to the WCS at any time by entering **UCS** in the command bar and choosing the **World** option.

Tip #128. Defining a user coordinate system (UCS) can involve anything from simply shifting the origin (0,0,0) to reorienting the axis so you can easily draw in a different plane. You can create as many user coordinate systems as you want, and then save and recall them as you need them.

Tip #129. Coordinates displayed in the status bar and by inquiry commands are relative to the current coordinate system — either a user coordinate system (UCS) or the World Coordinate System (WCS).

Tip #130. You use the **Plan** command to change to the plan view from any user coordinate system or the World Coordinate System.

Dimension Settings

Tip #131. There are hundreds of settings that affect how dimensions are created and displayed. To see how they work and to customize them, enter the **DimStyle** command.

Customizing and Programming

Tip #132. You can open a drawing (*.dwg* file), run a script (*.scr* file), or load a LISP program (*.lsp* file) by dragging it from the File Explorer and then dropping it on progeCAD.

When a *.dwg* file is dragged to progeCAD's title bar, it is opened as a drawing; when it is dragged into the current drawing, it is placed as a block.

Command Aliases

Tip #133. You can make aliases for any progeCAD command. *Aliases* are shortened command names. For example, you may prefer "guideline" over "infline." To make an alias, enter the **Customize** command, and then click the **Aliases** tab.
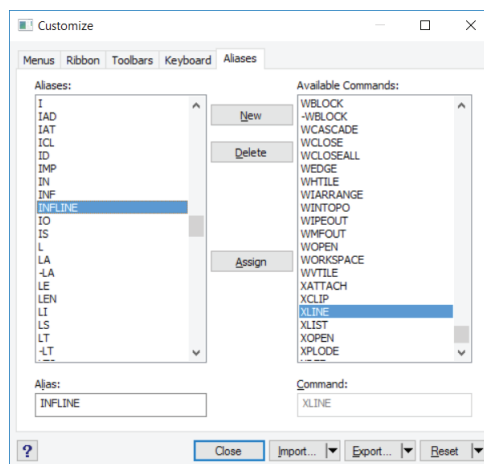
Tip #134. You can assign commands to any key on your keyboard or change existing key assignments. You can also use key combinations with **Shift**, **Alt**, and **Ctrl**. Enter the **Customize** command, and then click the **Keyboard** tab.
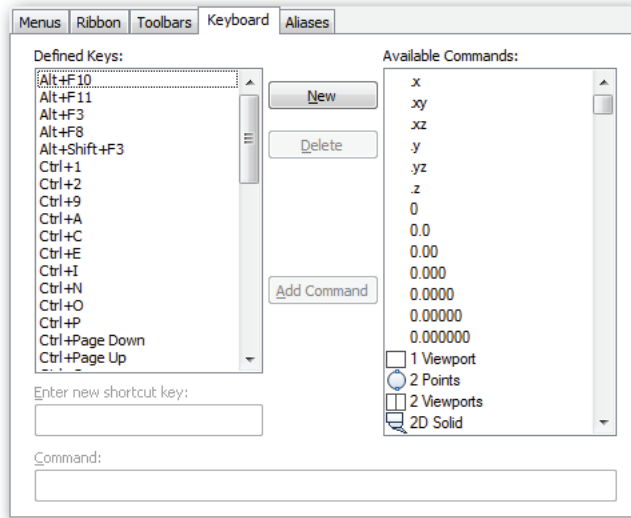
Recording Scripts

Tip #135. You record command entries and mouse coordinate selections in a script file, which you play back later. Doing this is handy for avoiding repetitive work. Scripts are saved in files with the *.scr* extension.

From the **Tools** menu, choose **Record Script**. When you have finished, choose **Stop Recording**.

Customizing Menus

Tip #136. You customize menus, toolbars, keyboard shortcuts, and aliases with the **Customize** command. To set them back to their original state, click **Reset**.

LISP

Tip #137. LISP is an interpreted programming language invented by John McCarthy in 1958 for work in artificial intelligence. You can write your own LISP programs for progeCAD that automate frequent or complex operations. See the online *progeCAD Developer Reference* for complete information.

You can enter a LISP expression in response to a command prompt. For example:

> Enter circumference: **(* 2 3.14 5)**

Tip #138. The **Multiple** command is not valid in LISP. See the *Developer Reference* section of Help for more information.

Tip #139. Any LISP function placed in the *icad.lsp* file is automatically loaded each time you start progeCAD.

Other Tips

Tip #140. progeCAD automatically saves your work at a time interval you specify. The default is every 15 minutes. Enter the **Options** command, choose the **General** tab, and then change the **Autosave** time.
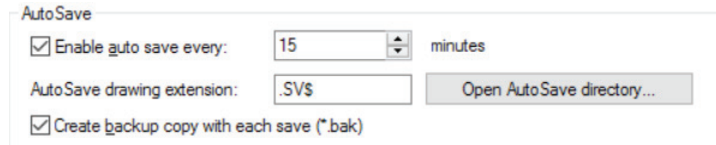
Tip #141. You can email a drawing to someone else while working in progeCAD. Enter the **eTransmit** command to package the drawing with all its support files.

progeCAD for AutoCAD Users

autoCAD is popular, but it expensive. Over its lifetime, Autodesk quadrupled the price to $4,195, and now it is available only by subscription, meaning users pay every year to use it, instead of just once. At time of writing, Autodesk charges $1,680/year; by contrast, progeCAD Professional has a one-time purchase price of $499.

And so we have a software package, progeCAD, that provides much of AutoCAD's drafting capabilities for a fraction of the cost. The cost/benefit ratio is attractive, especially as for many drafters much of what AutoCAD adds is inconsequential.

This chapter describes some of the similarities and differences between progeCAD and AutoCAD. Terminology between the two is virtually identical; the only significant difference is progeCAD's use of the words "esnaps" or "entity snaps" in place of AutoCAD's "osnaps" and "object snaps."

In This Chapter

- User Interface Comparison
- Command Name Differences
- Commands Enhanced in progeCAD
- Reading and Writing DWG Files
- Customizing and Programming progeCAD
- Commands and System Variables
- Additional Commands & System Variables
- AutoCAD Commands Missing from progeCAD
- progeCAD-AutoCAD Dictionary
- Folder Structure

AutoCAD 2017
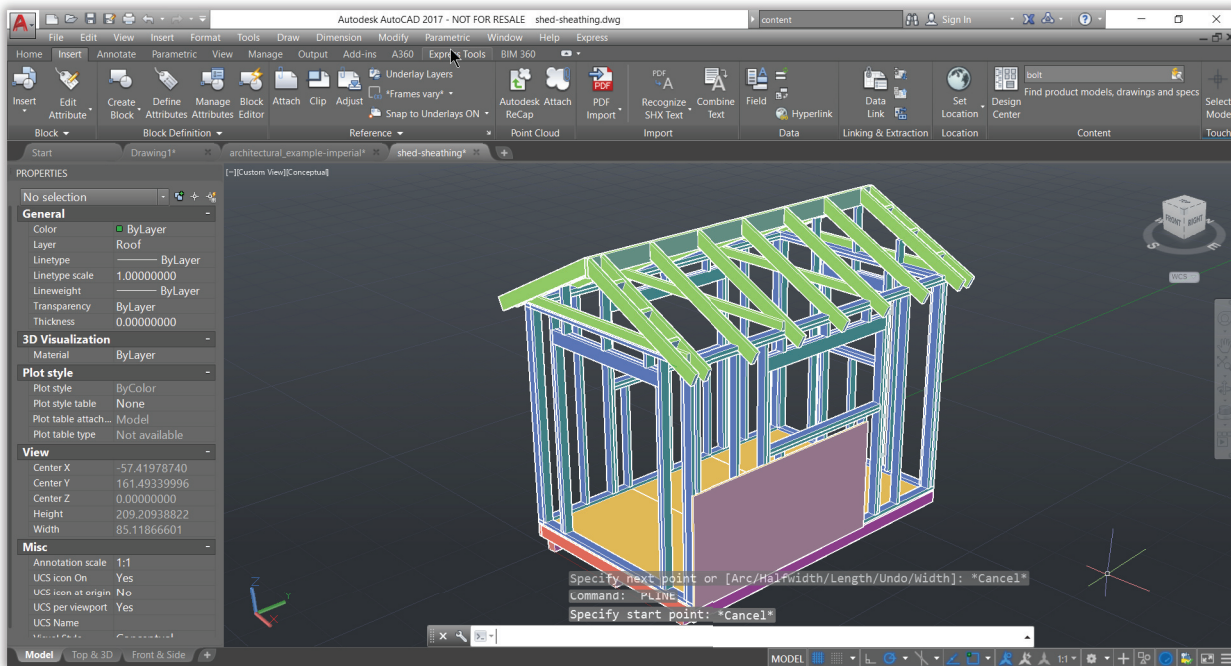
AutoCAD and progeCAD Share Similarities

Many aspects of AutoCAD and progeCAD are the same or very similar. Theses include the user interface, 3D and ACIS, paper space and layouts, and UCSs; the names of commands and system variables; the types of fonts, linetypes, and hatch patterns; the use of macros, scripts, LISP, and VBA programming; and basing drawings on *.dwg* and *.dwt* files.

For example, the names of commands, options, and system variables names are entered identically in both CAD packages. To draw a polyline in progeCAD, you enter the **PLine** command and press **Enter** — just as in AutoCAD. The prompt with its options looks identical, too:

> Command: **pline**
> Start of polyline: *(Pick a point)*
> Specify next point or [Arc/Distance/Follow/Halfwidth/Width]: *(Pick another point or enter an option)*

Some aspects, however, differ and that's what this chapter concentrates on. Both programs are constantly updated, and so these lists cannot be exhaustive.

progeCAD 2017
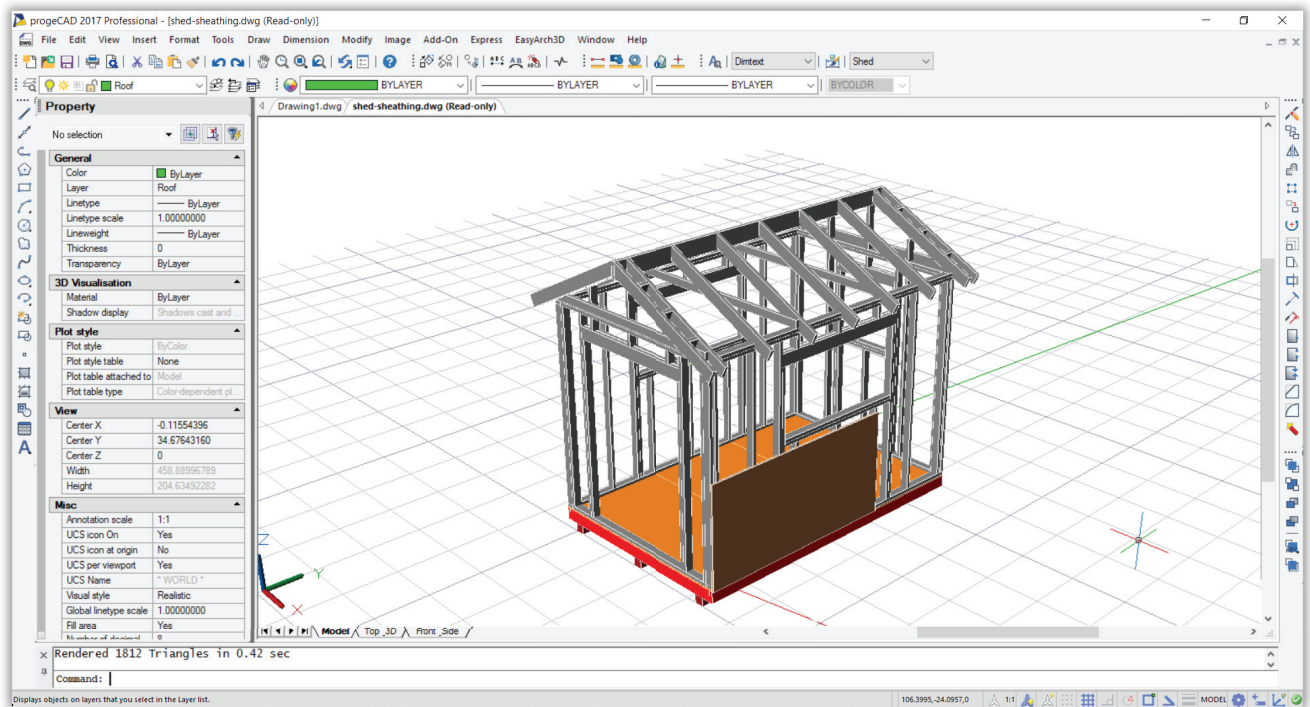
Highlighting progeCAD Advantages

While progeCAD is meant to be similar to AutoCAD in operation, the program nevertheless provides a package of functions to that give progeCAD an advantage. Here is a summary of some of progeCAD's special functions and how they compare with AutoCAD.

- Explorer
- iCADLib
- Cloud Storage
- Raster to Vector
- EasyArch 3D
- Google Earth
- Table Editing
- Audio Notes
- Express Tools

progeCAD Intro

The progeCAD Intro window is your gateway to learning more about the program and its add-ons.

To access it, enter the **ShowIntro** command. In AutoCAD, the equivalent is the Start tab.
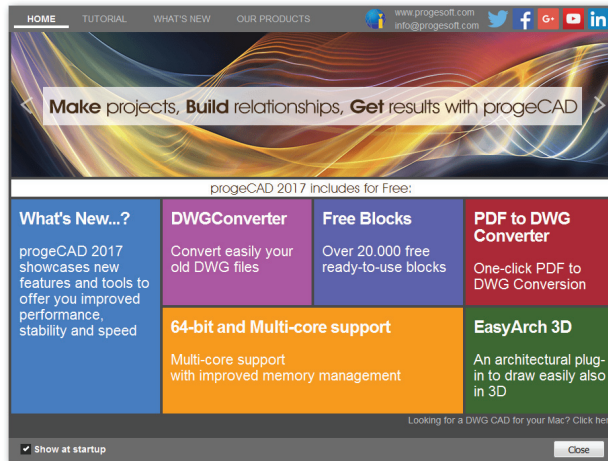
progeCAD Explorer

The progeCAD Explorer palette provides visual access to all design resources in drawings that are open — layers, layer states, linetypes, text styles, coordinate systems, views, layouts, blocks, dimension styles, groups, and external references. Being a palette, you can keep it open at all time.

To access it, enter the **Explorer** command. In AutoCAD, the equivalent is the DesignCenter palette.
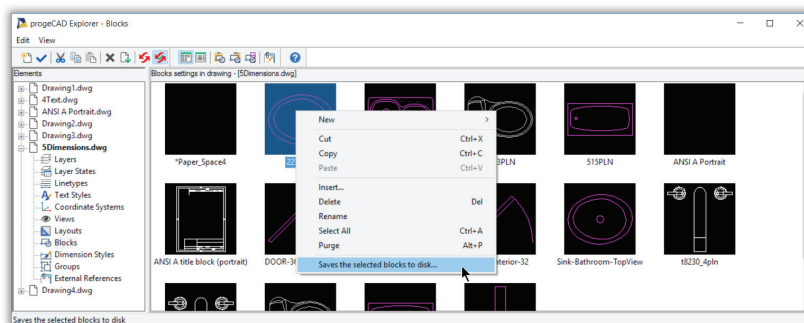
iCADLib

The iCADLib block library extends the Explorer to create libraries of blocks: specify the name of a folder with your blocks, and they are added as a library. In addition, progeCAD includes hundreds of 2D and 3D blocks free, which AutoCAD does not. iCADLib also accesses the mammoth part libraries hosted by Cadenas and Trace Parts. Being a palette, you can keep iCADLib open at all time.

To access it, enter the **IcadLib** command. In AutoCAD, the equivalent are the Content Server and an online parts library named Seek.
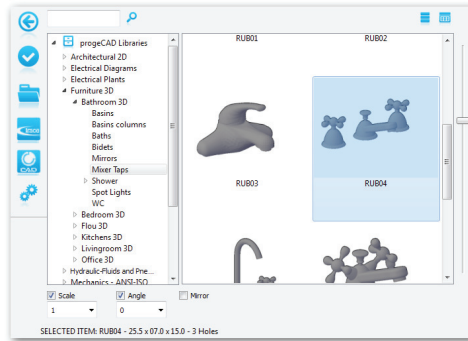
Cloud Storage

Cloud storage is a handy way of accessing your drawings when you are on the road, as well as giving other people access to drawings over the Internet — without sending them by email or mailing them on a USB thumbdrive. progeCAD supports a number of popular cloud storage providers, such as Dropbox, Google Drive, and Microsoft Onedrive. To use these services with progeCAD, they must be installed on your computers. Autodesk does not allow direct access to Dropbox and the other popular providers.

To open and save files from and to the cloud, enter the **OpenCloud** and **SaveCloud** commands. In AutoCAD, the equivalent is A360.
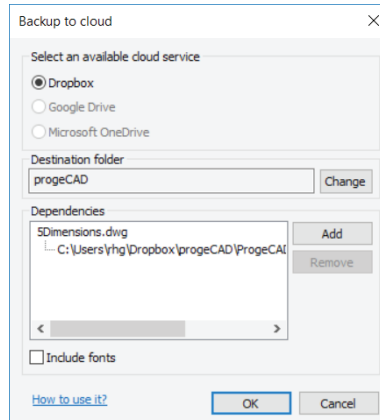
Raster to Vector Conversion

WinTopo converts raster images into vector entities. The images can be in common raster files, such as PNG, TIFF, and JPEG, or else captured from hardware like a scanner or digital camera attached to the computer. After conversion, the vectors can be saved in DXF and a half-dozen other formats. (To convert PDFs to vector, use progeCAD's PDF2CAD command.)

To access it, enter the **WinTopo** command. Autodesk charges for its raster-to-vector conversion software.
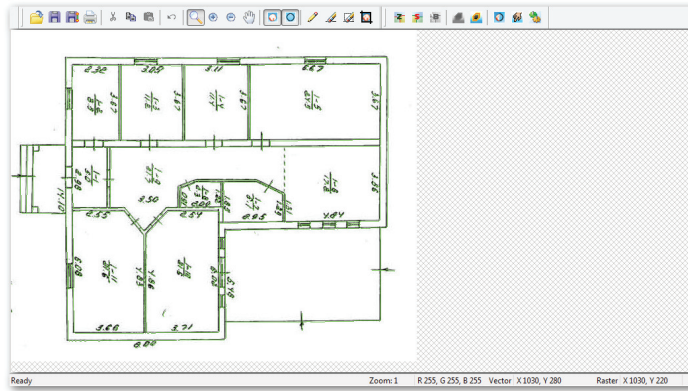
EasyArch 3D

The EasyArch 3D add-on allows you to create architectural drawings in 2D (floor plans) and 3D. It provides commands for drawing walls with automatic cleanup, inserting parametric doors and windows, place stairs and balconies, add roofs, and do area calculations.

You access EasyArch 3D's command through the EasyArch3D menu or ribbon tab. AutoCAD charges extra for its architectural design software.
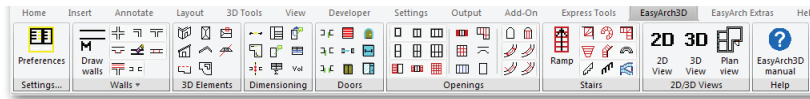
Google Earth

The Google Earth add-on lets you place "tiles" (mapping images) from Google Earth in drawings appropriate to the location of the model. This lets you see the project in the context of its surroundings. Conversely, you can upload models to Google Earth as progeCAD converts them to KML format.

To access mapping images and save models to Google Earth, enter **ImportGeImage** and **PublishKml** commands; they require that Google Earth is installed on your computer. AutoCAD includes a mapping service, but it requires that you have an account with A360.
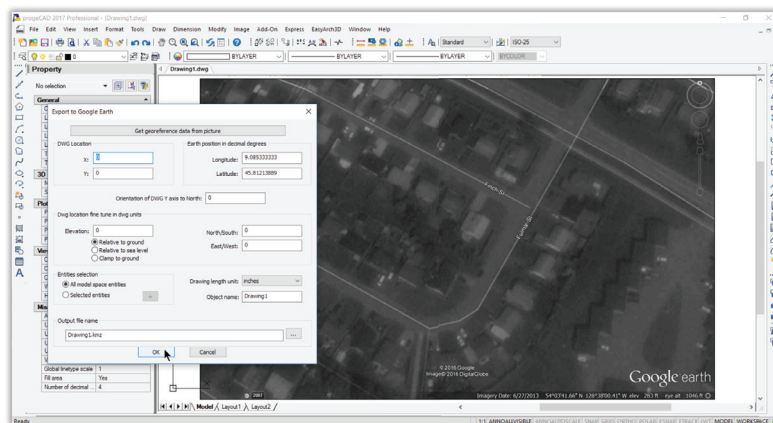
Table Editing

Not only does progeCAD provides many of the commands found in AutoCAD, sometimes it also extends the commands with additional useful functions. For example, the **Circle** command's **Arc** option extends the angle of an arc to a full 360 degrees, making it a circle; the Circle command in AutoCAD does not do this.

Especially impressive are the additions progeCAD made to the **TablEdit** command. Editing a cell presents a dialog box with cursor keys for easy navigation between cells, as well as all options for formatting cells. AutoCAD provides the cell formatting options, but not the easy cell navigation.
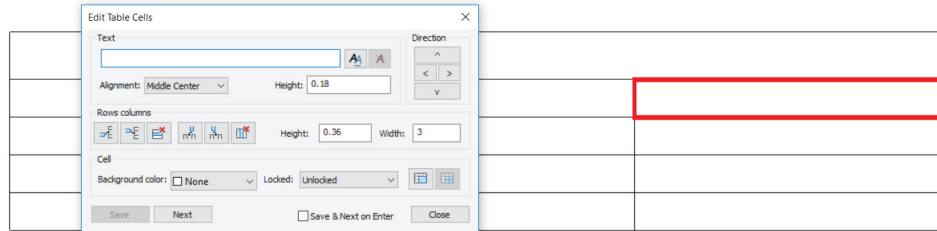
Audio Notes

It can be useful to attach voice messages to drawings, particularly when working in the field. progeCAD lets you do this through its Audio Note function, which records, plays back, or attached WAV audio files.

Attach audio notes with the **AudioNote** command. AutoCAD does not offer this function.
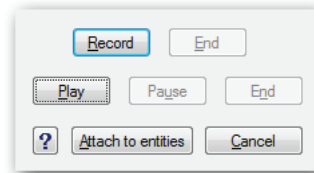
Express Tools

progeCAD offers an extra set of commands that perform unusual functions and so are not used as often. These include placing text along an arc, placing a break line, and exploding text into entities like lines and arc.

You access Express Tools through the Express menu or Express Tools ribbon tab. AutoCAD offers a similar set of Express tools.
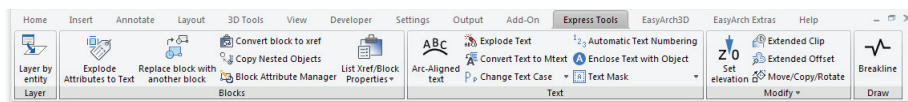
Comparing Commands and System Variables

For the most part, commands, options, and system variables operate identically in both CAD packages. There are some differences between the two, and this section describes them.

Command Differences

The core commands between progeCAD and AutoCAD are identical. Each CAD program seeks to differentiate itself from competitors, and so progeCAD includes an additional set of commands not found in AutoCAD. Some progeCAD commands have names that differ from those in AutoCAD. Sometimes progeCAD has commands where AutoCAD has only keystrokes; for example, AutoCAD has no Coordinate command like progeCAD has, but provides users Ctrl+D and F6 instead.

progeCAD	AutoCAD	Action in progeCAD different from AutoCAD
audionote	No equivalent	Creates and plays audio notes attached to entities.
cmdbar	No equivalent	Positions the command window.
coordinate	Ctrl+D *or* F6	Changes the display of coordinates on the status line.
copyedata	No equivalent	Copies extended entity data from one entity to another.
customize	cui	Displays the Customize dialog box.
editedata	No equivalent	Edits extended entity data found in an entity.
esnap	-osnap	Sets entity snaps from the command line.
expblocks	insert	Displays the progeCAD Explorer - Blocks.
expdimstyles	ddim	Displays the progeCAD Explorer - Dimension Styles.
expfonts	style	Displays the progeCAD Explorer - Styles.
explayers	layer	Displays the progeCAD Explorer - Layers.
explorer	rename	Displays the progeCAD Explorer.
expltypes	linetype	Displays the progeCAD Explorer - Linetypes.
expucs	dducs	Displays the progeCAD Explorer - UCS.
expviews	view	Displays the progeCAD Explorer - Views.
moveedata	No equivalent	Moves extended entity data from one entity to another.
mslide	mslide	Also makes EMF and WMF files of the current view.
qprint	No equivalent	Quickly prints the current viewport with no options.
reassocapp	No equivalent	Reassociates extended entity data with an application.
recscript	No equivalent	Starts the Script Recorder.
rtrotx	No equivalent	Rotates the view of entities while maintaining the x-axis.
rtroty	No equivalent	Rotates the view of entities while maintaining the y-axis.
scrollbar	No equivalent	Toggles the scroll bars
selgrips	No equivalent	Select objects to display grips
statbar	No equivalent	Toggles the status bar
stopscript	Esc	Stops running the script.
vmlout	No equivalent	Exports drawings in VML format
vslide	vslide	Also displays EMF and WMF files in the current viewport.

Additional progeCAD System Variables

To provide greater flexibility in configuring the program, progeCAD includes additional system variables that are not found in AutoCAD. Some of theme include the following ones:

Command	Action in progeCAD
AutoMenuLoad	Determines whether default menu or menu associated with drawing is loaded
BaseFile	Name of default prototype drawing used to create new drawings
CmdLnText	Specifies the command bar prompt
ColorX	Color of x axis in the cursor and UCS icon
ColorY	Color of y axis in the cursor and UCS icon
ColorZ	Color of z axis in the cursor and UCS icon
ExplStBlk	Column width of block settings in Explorer
ExplStLay	Column width of layer settings in Explorer
ExplStLt	Column width of linetype settings in Explorer
ExplStTs	Column width of text styles settings in Explorer
ExplSstUcs	Column width of UCS settings in Explorer
ExplStVw	Column width of named view settings in Explorer
IcadVer	Reports the progeCAD version number
LastCmdPt	Stores the last point entered relative to the current UCS
LastPrompt	Stores the last prompt string displayed by command bar
LtSclWblk	Specifies whether global linetype scale is applied to inserted blocks
MacroRec	Toggles macro recorder
MacroTrace	Toggles macro tracer
MakeBak	Renames drawing as .bak with each save
NFileList	Number of most recently used files in File menu
ProgBar	Toggles display of the progress bar
PromptMenu	Toggles display of the prompt menu
SaveRoundtrip	Determines whether unknown entities are saved R14 format
ScrlHist	Specifies number of command lines to track in command bar
SrchPath	Specifies the search path for support files
TbColor	Switches toolbar color between color and monochrome
TbSize	Switches toolbar size between small and large
Tipofday	Toggles Tip of the Day dialog box
ViewAspect	Aspect ratio of the view
WndBtExp	Specifies width of the list pane in Explorer
WndLCmd	Toggles command bar visibility
WndLMain	Toggles display of the main window
WndLMdiW	Controls display of MDI child window
WndLStat	Toggles display of the status bar
WndLtExp	Toggles display of the Explorer window

WndLText	Toggles display of the prompt history window
WndPCmd	Screen coordinates of the command bar when floating
WndPMain	Screen coordinates of drawing window position
WndPPmenu	Screen coordinates of the prompt menu
WndPtExp	Screen coordinates for Explorer window
WndPText	Screen coordinates for prompt history window
WndSCmd	Size of command bar, when floating
WndSMain	Size of the drawing window
WndStExp	Size of the Explorer window
WndSText	Size of the prompt history window
ZoomPercent	Incremental zoom percentage

Commands Enhanced in progeCAD

Going along with the theme of providing greater value to users, some of progeCAD's commands provide more options than do those in AutoCAD. Here is a selection of them.

Command	Action in AutoCAD	Additional Action in progeCAD
Circle	(Use Join command)	**Arc** option converts arcs to circles
Line	(Use direct distance entry)	**Angle** option specifies angle, followed by a length **Length** option specifies length, followed by an angle.
MSlide	Saves screen Autodesk's proprietary SLD format.	Also saves in Microsoft's WMF (windows meta file) and EMF (enhanced meta file) formats
Offset	Copies to one side at a time	**Bothsides** option copies to both sides at once
Solid	Draws filled quadrilaterals and triangles.	**Rectangle** option draws solid-filled rectangles **Plane Square** draws solid-filled squares **Plane Triangle** draws solid-filled triangles

Expanded Capabilities in progeCAD

In addition to the larger number of options, progeCAD provides users with additional flexibility in how some commands are carried out. Here are some of these:

- progeCAD's entity (object) snaps can be entered directly at the command prompt. AutoCAD's requires that its **-OSnap** command be employed as a type of prefix.
- progeCAD uses the + and - keys to add and remove objects from selection sets.
- progeCAD's **Select** command has additional selection options not found in Auto-CAD:

Select	Meaning	Comment
CC	Crossing Circle	All entities within and crossing a circular area
O	Outside Window	Selects all entities outside a rectangular window
OC	Outside Circle	Selects all entities outside a circular area
OP	Outside Polygon	Selects all entities outside a polygonal area
PRO	Properties	Selects all entities with specific properties
WC	Window Circle	Selects all entities within a circular area

AutoCAD Commands Missing from progeCAD

Each release of AutoCAD adds more commands, some of which progeCAD also adds, some of which progeCAD already boasted, and some of which are not added. For example, AutoCAD 2017 added the ability to import vector entities and embed TTF fonts from PDF files; progeCAD already did this. On the other hand, progeCAD does not import point cloud data from 3D laser scanners, because most users have no need to works with billions of points in 3D space.

Reading and Writing DWG Files

progeCAD reads and writes drawings created by AutoCAD through to 2017. Because progeCAD does not support everything in AutoCAD, some objects are difficult to handle. Tables, for instance, are displayed but as anonymous blocks and cannot be edited in progeCAD.

The addition of ObjectARx to AutoCAD Release 13 enabled Autodesk and third-party developers to create "custom objects" that can be created and edited only when the associated ObjectARx programming code is present. When the code is missing, then a *proxy* object is displayed instead. Autodesk provides *object enablers* that provide sufficient code to edit custom objects. progeCAD converts proxy entities into blocks using their exclusive proxy conversion technology built into the software.

In general, AutoCAD imports and exports a larger number of formats than progeCAD, with a couple of notable exceptions:

- progeCAD reads and converts *.dwf* files, but AutoCAD does not.
- progeCAD and AutoCAD both read, write, and convert vector data in 2D *.pdf* files, but AutoCAD does not export 3D models in 3D PDF format.

Customizing and Programming

progeCAD works with most of the support files used by AutoCAD, including *.lin*, *.pat*, *.mnu*, *.fmp*, *.unt*, *.dcl*, *.lsp*, *.shx*, and *.sld* files. progeCAD reads and writes AutoCAD's *.cui* menu files.

progeCAD mimics AutoCAD in writing menu and toolbar macros, scripts, DCL (dialog control language), LISP routines, i-drop, and VBA applications (visual basic for applications).

Unlike present-day AutoCAD, progeCAD can work with programs that use the SDS API (solutions development system applications programming interface), called "ADS" in AutoCAD until it was removed with Release 13.

AutoCAD's rarely-used side-screen menu is not included in progeCAD. progeCAD's prompt menus replace some of the functions provided by the side-screen menu.

progeCAD also works with Diesel, the pseudo-programming language used with macros and for customizing part of the status bar.

progeCAD-AutoCAD Dictionary

This brief progeCAD-AutoCAD dictionary explains some progeCAD jargon:

progeCAD Term	AutoCAD Equivalent
command bar	command prompt area
edit length	lengthen
entity	object
entity snap	object snap
entity snap precision	aperture
edata	xdata
fixed attribute	constant attribute
follow	continue
freehand	sketch
hidden attribute	invisible attribute
infinite line	xline
insert	draw
orthogonal	ortho
parallel	offset
plane	solid (2D)
point snap	node snap
predefined attribute	preset attribute
prompt box	context menu
prompt history window	text screen
reference grid	grid
snapshot	slide
validate attribute	verify attribute

Introducing the Boardwalk

to help you learn progeCAD, this book walks you through a fictional condominium design project called "The Boardwalk on 18th Avenue." For this project, you are to produce the floor plans for one-bedroom condo units.

The design project goes through the following stages:

1. Draw the *exterior walls*; these enclose the unit.
2. Define the *interior spaces*; these include the living room, bedroom, and closets.
3. Determine the *access points and views*, such as doors and windows.
4. Specify the *fixed amenities*, such as bathroom fixtures, kitchen arrangement, and fireplace. Also specify the *non-fixed amenities*, such as living room seating, dining table, and bed.
5. Label important areas with text.
6. Add border and title block.
7. Apply dimensions to show sizes.
8. Create a printout of the floor plan drawing.

The following chapters describe how to work on the drawings for each stage using progeCAD.

About The Condominium Project

The Boardwalk on 18th Avenue is a low-rent, one-story condominium development consisting entirely of one-bedroom apartments. A *condominium* is an apartment or flat that is purchased, and not rented or leased. The development takes up half a city block, and so is surrounded on three sides by streets.

Development on the site is limited by the city's bylaws, which define the *building lines* and parking requirements. No part of the structure can be beyond the building lines; a pair of units are positioned in the illustration below. (P/L indicates the property lines.)
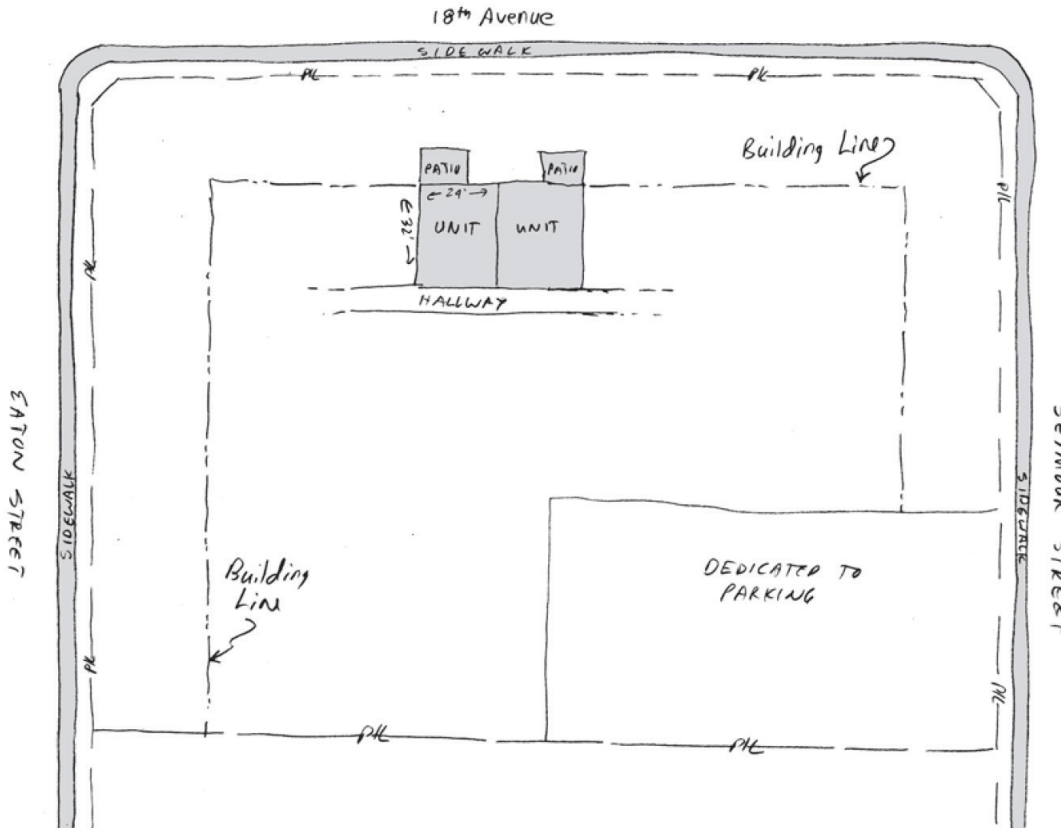
Each unit is the same: 32 feet deep by 24 feet wide (768 square feet). Inside the unit, the layout is *open plan*, meaning the kitchen, dining room, and living room are open to one other, with no full walls between.

Each unit consists of the following items:

- Kitchen.
- Dining room.
- Living room with fireplace.
- Three-piece bathroom with sink, toilet, and bathtub.
- Bedroom with closet and ensuite bathroom. An *ensuite* has a sink and toilet, and is accessed only from a bedroom.
- Hallway with coat closet and laundry closet with hot water tank.

Residents access their units from an outside door or from the central hallway. Outside the living room is a 10' x 14' concrete slab patio.

A rough sketch of the layout of a unit is illustrated below.
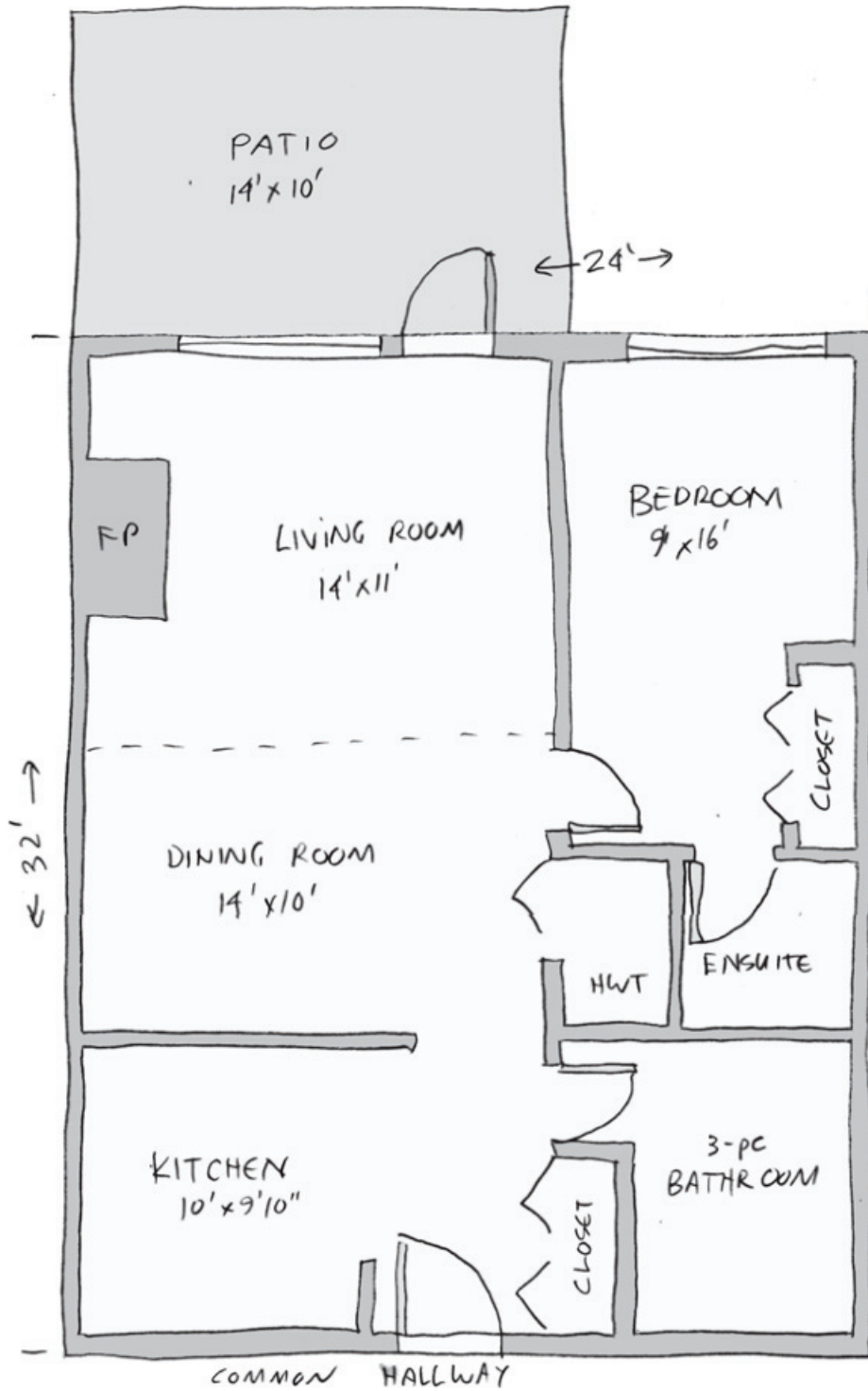
Every second unit is a mirror image of this one, so that similar facilities adjoin one other: bathrooms with their plumbing needs are adjacent; (quieter) bedrooms have common walls, as do the (nosier) living rooms and kitchens.

As the drafter for this project, your task is to draw the 2D floor plan of the unit. You use *layers* to segregate information, and *blocks* to place common objects, such as doors and chairs. You must place *dimensions* to show the size of important parts of the drawing, and add *text* to explain functions. You *plot* the drawings to scale so that your client can review them.

The completed progeCAD drawing should look something like this:
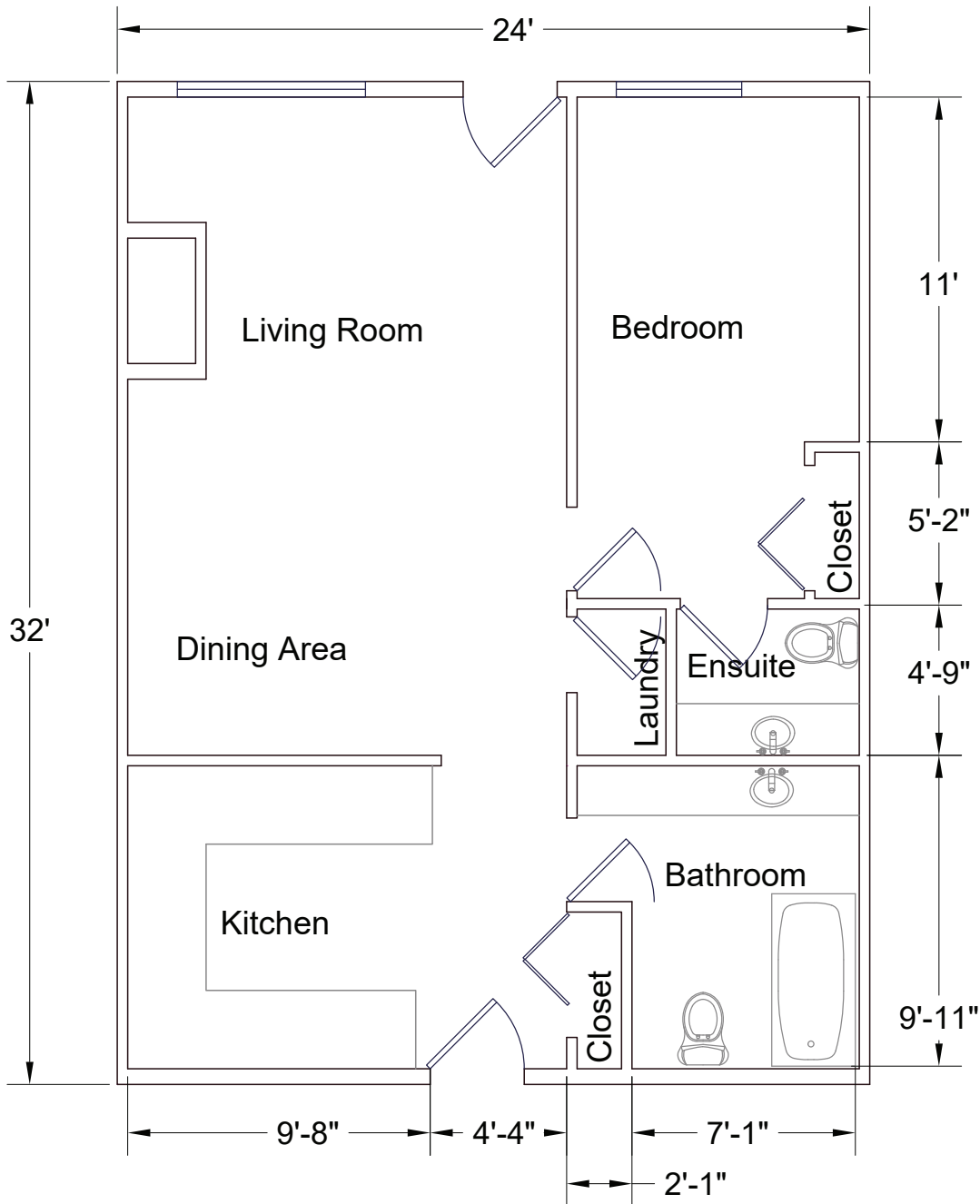
Setting Up New Drawings

before drawing even one line, you must first prepare new drawing files with suitable settings. For this tutorial, you are creating a medium-size architectural drawing, and so in this chapter you prepare the new drawing file for architectural drafting.

To help you prepare the drawing, you use two sets of tools:

- New Drawing Wizard sets up some parameters.
- Additional commands set up the remaining parameters.

In This Chapter

- New Drawing Wizard
- Setting Limits
- Toggling Ortho Mode
- Calculating Scale Factors
- Naming (and Saving) Drawings
- Specifying Snap & Grid Spacing
- Selecting Entity Snap Modes
- Alternative Method: Settings Dialog Boxes

New Drawing Wizard

To help you set up new drawings, progeCAD provides the New Drawing wizard. *Wizards* are in general multi-page dialog boxes that take you through a complex process. The New Drawing Wizard takes you through the steps to specify settings for new drawings. It is part of the Start dialog box.

TIP   The settings you make in progeCAD are never permanent; you can always change them at any time later. Wizards do not, unfortunately, provide all the settings needed, so sometimes you need to set additional options later.

To use the New Drawing wizard for setting up drawings for an architectural style of drafting, follow these steps:

1. Start progeCAD. Notice that it opens with a blank, new drawing.

2. Access the Start dialog box (and then the New Drawing Wizard,) using one of the following methods:

   • Type the **PNew** command at the 'Command:' prompt.

   • Or, at the menu bar click **File** and then select **New**.
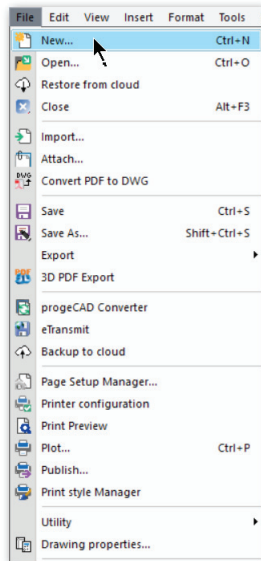
   • Or on the Standard toolbar, click the **New** button.

   • Or in the ribbon interface, click the **New** button in the quick access toolbar.

3. Notice that the Start dialog box presents a variety of ways of starting a new drawing:

To summarize them quickly here:

- **Start from Scratch** uses the default metric or imperial template drawing as the basis of the new drawing
- **Use a Template** lets you choose which template drawing to use for the new drawing
- **Use a Wizard** lets you specify a number of settings for the new drawing

Each option lets you have more input over the content of the new drawing at the cost of taking longer to start. For this tutorial, choose the **Use a Wizard** button, and then click **Next**.

4. Notice that the first page asks you about the foundation for the new drawing. It presents you with two choices:
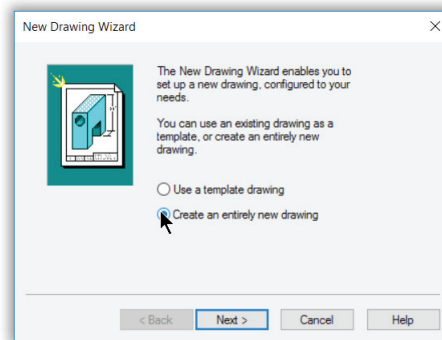
○ **Use a template drawing** — the new drawing is based on an existing drawing known as a "template." progeCAD copies all the settings from the template file. This option is a good pick when your office has already created *.dwt* template files.

(If you were to choose this option, the wizard's next page would ask you to select the name of the template file.)

⊙ **Create an entirely new drawing** — the new drawing is based on *icad.dwt*, a blank drawing that contains progeCAD default settings.

Select the **Create an entirely new drawing** option, and then click **Next**.

5. Click **Next**, again.
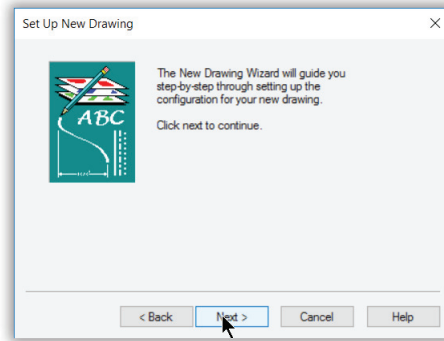
6. progeCAD works with metric or imperial units. A lot of settings depend on which one of the two is in effect, and so progeCAD shows you a table of them.
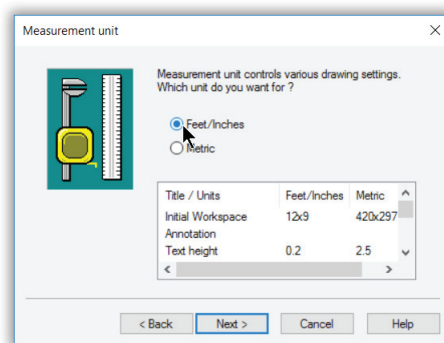
For this tutorial, choose **Feet/Inches**, and then click **Next**.

7. progeCAD lets you work with five types of linear measurement units. "Linear" refers to distances, as opposed to angles.
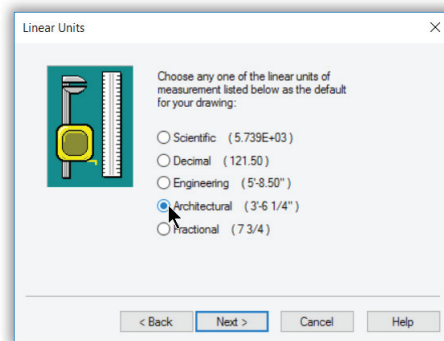
The units of linear measurement available to you are the following:

- **Scientific** – displays scientific notation, such as 5.739E+02
- **Decimal** – displays unitless decimals, such as 573.9; this format is used for drawings with metric units.
- **Engineering** – displays feet and decimal inches, such as 5'-7.39"
- **Architectural** – displays feet and fractional inches, such as 5'-7 3/4"
- **Fractional** – displays fractional inches only (no feet), such as 67 1/4"

Select **Architectural** units, and then click **Next**.

8. In the same way, progeCAD lets you work with a variety of angular units.
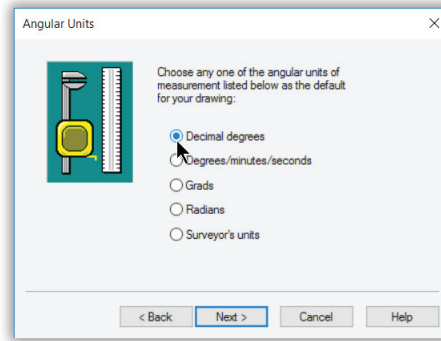
The units of angular measurement available to you are the following:

⊙ **Decimal degrees** – displays degrees and decimals, such as 5.739°

○ **Degrees/Minutes/Seconds** – displays degrees with minutes, seconds, and decimals of seconds, such as 5° 73' 29.33"

○ **Grads** – displays grad and decimals, such as 234.55g; there are 400 grads in a circle, and grads are used in Germany.

○ **Radians** – displays radians and decimals, such as 57.34r; there are 2*pi radians (approximately 6.282 of them) in a circle, and radians are used by scientists and programmers

○ **Surveyor's Units** – displays angles relative to North, such as N57d03'29"E

Select **Decimal degrees**, and then click **Next**.

9. The next step is to decide whether you want to used *named style tables.*
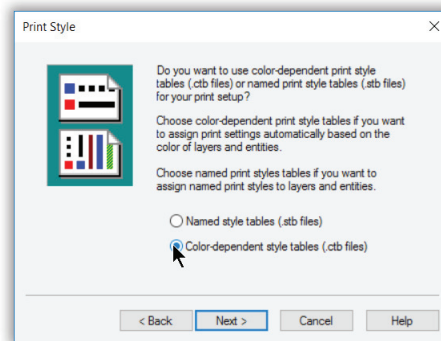
Styles allow you superb control over the plotted look of every entity in the drawing. Both files — *.stb* (short for "style table") and *.ctb* ("color table") — store settings; both are compatible with AutoCAD.

○ **Named style tables (.stb files)** – controls the look of every entity, such as thickness, grayscale, line ends, and more; applies named styles to entities and layers

⊙ **Color-dependent style tables (.ctb files)** — only the color of entities can be used to affect the plotted output

For this tutorial drawing, the older, color-dependent style tables are just fine. Select **Color-dependent style tables**, and then click **Next**.

Alternative Method: Settings Dialog Box

To set up a new drawing, there is an alternative to the New Drawing wizard: using the Settings dialog box. It controls all aspects of how a drawing looks. To access it, enter the **Settings** command. Notice that it has several tabs.

Drawing Units Tab

The Drawing Units tab specifies settings for linear units, angular units, and user-defined coordinate systems. Select among these categories from the **Change Settings For** droplist.
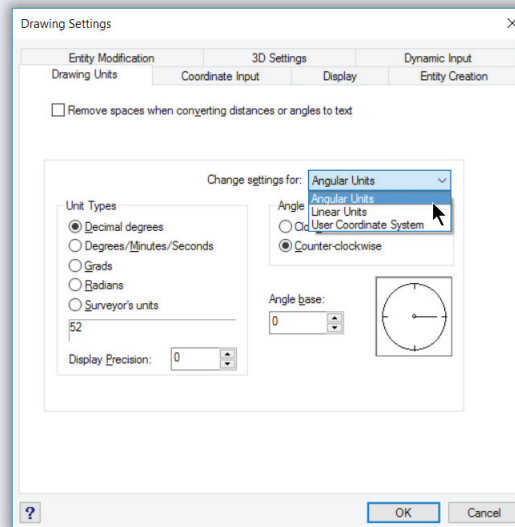
Coordinate Input Tab

The Coordinate Input tab specifies settings for snap and grid, entity snaps, entity selection, limits, and polar tracking. Select among these categories from the **Change Settings For** droplist.
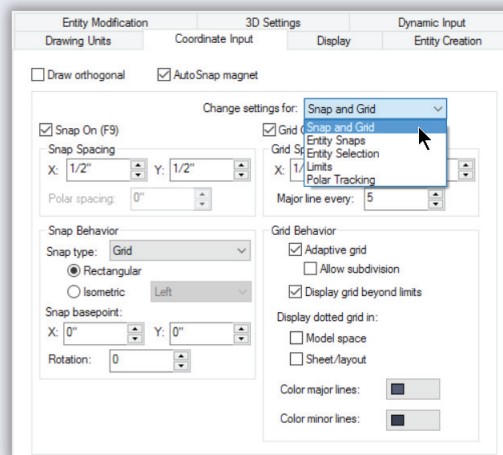
10. The final page of the wizard calls itself "entity creation," but deals with more than just that.
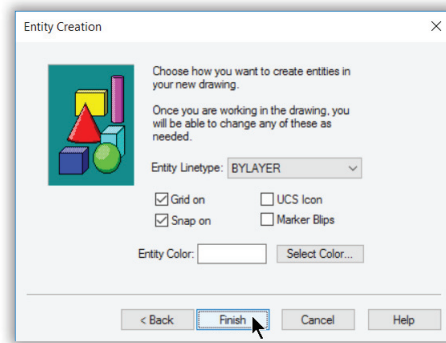
The meaning of the options are as follows:

Entity Linetype specifies how entities are assigned linetypes. This is an odd question to ask when defining a drawing globally, in my opinion. In any case, always select **ByLayer** from the options in the droplist:

- **ByLayer** means that layers define the linetype for all entities residing on that layer.

- **ByBlock** means that blocks define the linetype for all entities in the block.

- **Continuous** means that entities are defined on a by-entity basis. In this case, the default is continuous (unbroken) lines.

☑ **Grid on** displays the grid, a matrix of dots that help you visualize relative sizes in drawings. Turn this option **on**.

☑ **Snap on** turns on the cursor increment, the distance the cursor jumps. Instead of moving freely, the cursor can be made to move in increments of one inch, a foot, a quarter-inch, whatever. Turn this option **on**.

☐ **UCS Icon** displays the UCS (user-defined coordinate system) in the drawing. This is useful for determining your 3D position in space, but is pointless for 2D drawings. Turn this option **off**.

☐ **Marker Blips** displays small + marks each time you click a point in the drawing. These blips can be helpful to determining where you clicked, but after a while clutter the drawing with otherwise-useless markers. (You clean the blips off drawings with the Redraw command.) Turn this option **off**.

Entity Color specifies the default color for entities in the drawing. As with linetypes, I find this an odd question; the best option is **ByLayer**. If you want, however, you can click the **Select Color** button and pick a default color. You have these two options:
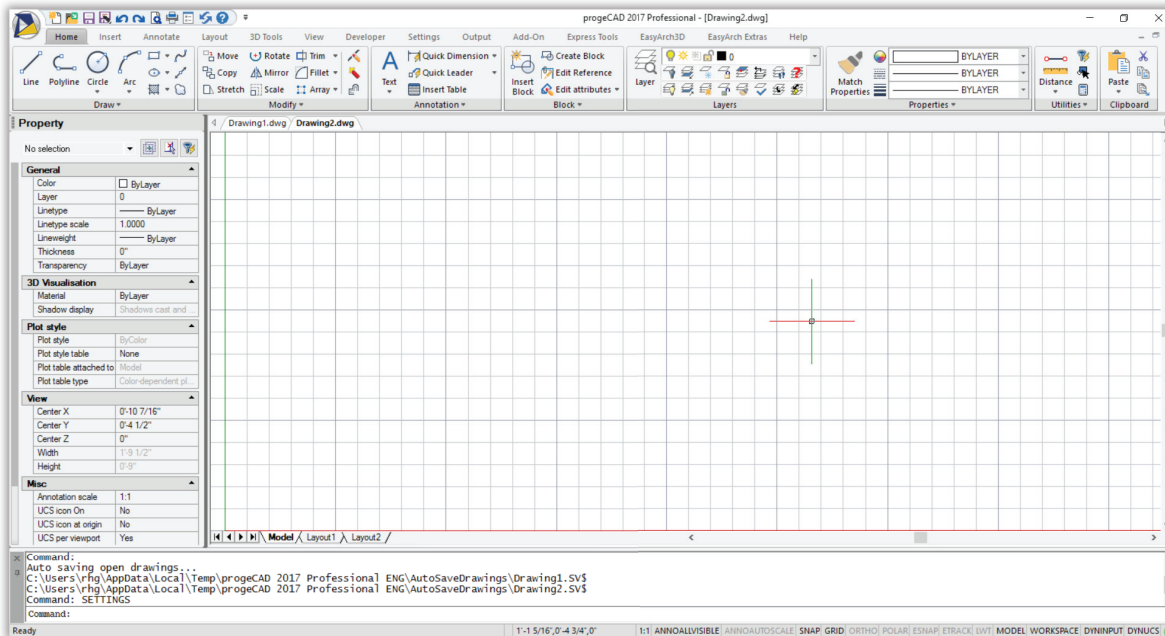
- **ByLayer** means that layers define the color for all entities residing on that layer.

- **ByBlock** means that blocks define the color for all entities in the block.

11. Click **Finish**. progeCAD applies the settings you selected to the drawing.

Notice that the grid is turned on and that the UCS icon is missing. On the status line, the SNAP and GRID settings are in black, meaning they are turned on. (If you cannot see the grid, use the **Zoom Extents** command, as follows:

    : **zoom**
    In/Out/All/Center/Extents/Left/Previous/Right/Window/Dynamic/<Scale (nX/nXP)>: **e**

There are more settings that every new drawing needs that are missing from the wizard. They include a scale factor, limits, ortho, snap and grid spacing, and the file name for the drawing. We'll tackle them after you save and name this drawing.

<u>SAVE COMMAND</u>

Saves drawings by name.

Command: **save**

Alias: **sa**

Menu: **File | Save As**

<u>ALTERNATIVES</u>

QSave saves without prompts.

Exports saves in other formats.

Naming (and Saving) Drawings

At this point, the drawing is devoid of entities, but still contains data. It is crucial that you save your work so that you don't lose fruits of your labor when Windows crashes your computer. In addition to you manually saving your work, the good news is that progeCAD automatically saves your drawing every 15 minutes.

The first time you save a new drawing, progeCAD asks you to give it a name. After that, the drawing is saved by the same name — until you choose to change it.

Use the **Save** command to save drawings and give them a name.

1. Enter the **Save** command at 'Command:' prompt.

2. The Save Drawing As dialog box presents numerous options:

Save in specifies the name of the drive and folder in which drawings are to be saved. You can also access *networked drives*, which are drives on computers connected to yours through a network. Click the **Save In** droplist, and then select a drive and folder for this drawing.

☑ **Use Preview** creates a preview image of the drawing. This is useful when browsing through drawings, in deciding which one to load. Turn this option **on**.

File name identifies drawings by names of up to 255 characters. For this tutorial, use the generic name "apartment."
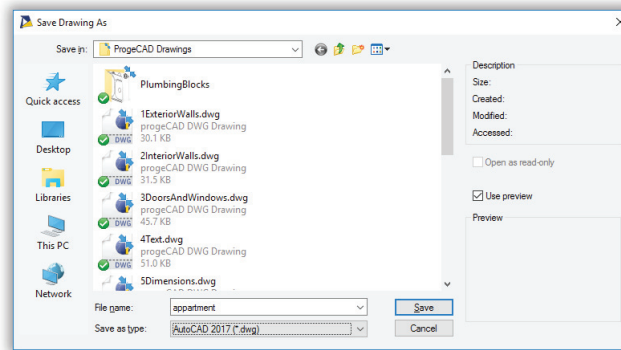
Save as type specifies the format in which to save the drawing. progeCAD supports the following AutoCAD-compatible formats:

*   DWG (standard drawing format)
*   DXF binary (drawing interchange format)
*   DXF ASCII
*   DWT (template drawing format)
*   DWG and DXF formats compatible with old releases of AutoCAD and progeCAD.
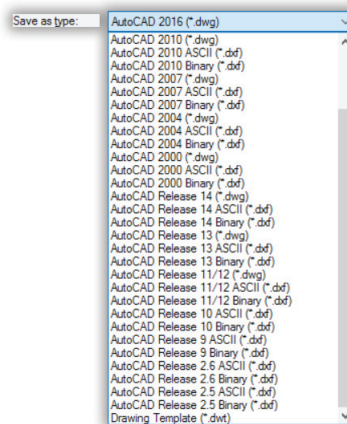
Unless you understand the reason for using another format, select **AutoCAD 2017 (*.dwg)**.

3. Click the **Save** button to save the drawing, giving it the new name. You should see the new file name on progeCAD's title bar.
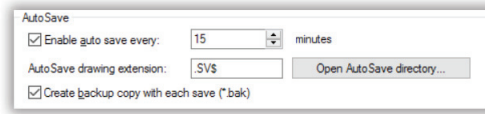
Setting Limits

Limits define the extents of the drawing: how big is the drawing horizontally and vertically?

LIMITS COMMAND

Sets grid and zoom-all limits.

Command: **limits**

Menu: **Tools | Drawing Limits**

In practice, limits don't actually matter, because progeCAD can create drawings as large as our solar system. (Printing a drawing that size might be a problem!)

Limits are useful because they define the limits of the grid and the zoom-all display. This provides you with an idea of the working area. For this tutorial, the limits of the drawing is the size of the condominium apartment, plus some extra space around the edges. The apartment is 32' x 24', so make the limits 40' x 30'.

Limits are set with the **Limits** command.

1. Enter the **Limits** command.

2. In the command bar, progeCAD asks you for the coordinates of the lower left corner. They are at 0",0" so just press **Enter** to accept the default values:

   Command: **limits**

   Limits are off: Specify lower left corner or [ON] <0'-0",0'-0">: *(Press **Enter**)*

30',40'

0,0 (the drawing origin)

3. The upper right corner uses the coordinates specified earlier. Remember to include the apostrophe (') after each dimension, otherwise progeCAD thinks you are entering inches.

> Upper right corner <0'-12",0'-9">: **40',30'**

4. Use the **Zoom Extents** command to "see" the limits of the drawing. progeCAD most likely complains, "The grid spacing is too small to display in this view." You fix the problem in the next section.
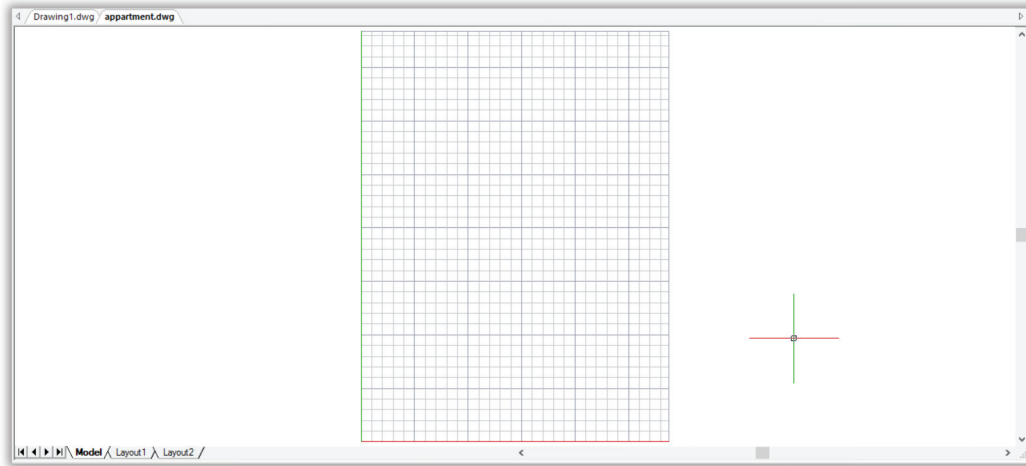
TIP   If the grid covers the entire drawing area, instead of being restrained to the limits, then make this change in the Settings dialog box:
1. Enter the **Settings** command and then choose the **Coordinate Input** tab.
2. If necessary, select **Snap and Grid** from the "Change Settings For" dropbox.
3. In the "Grid Behavior" area, turn off the **Display Grid Beyond Limits** option.
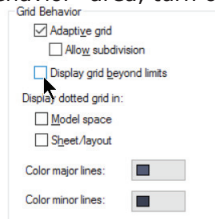
4. Click **OK**. The grid now displays correctly.

Specifying Snap and Grid Spacing

The New Drawing wizard had preset the spacing between snap points and grid dots to 1/2".
But you can change this, and here's how.

Changing the Grid Spacing

When you zoom in and out of the drawing, progeCAD change the number of grid lines it displays, otherwise the screen can end up looking black from too many lines.

I prefer to have a fixed spacing, however, so that it doesn't change on me. This is done with the **Grid** command. Change the grid spacing to 1', as follows.

1.   Enter the **Grid** command, or else right-click GRID on the status bar, and then select **Settings** from the shortcut menu.
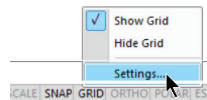
2.   In the command bar, enter the spacing of 1':

Grid is on: Specify grid spacing or [OFF/Snap/Aspect]<1/2">: **1'**

Each grid line is now one foot from its neighbor, although visually the grid might not look different to you.

Changing the Snap Spacing

The *snap spacing* determines how far the cursor moves. You can think of it as cursor resolution. When you draw the floor plan of this apartment, you will probably draw entities to the nearest 1/2" or so. By setting snap to 1/2", you make it easier to pick points with the cursor.

Snap spacing is related to grid spacing. In fact, the two can share the same value. In practice, I find that the grid should have a larger value (like 1 foot or 1 meter), and the snap a smaller on, like 1/2" or 1 mm.

The snap spacing already had been set to 1/2" by the New Drawing Wizard. If you wish to confirm this, do so with the **Snap** command.

Toggling Ortho Mode

The drawing of the apartment is *rectilinear*, which means it consists mostly of straight lines that are drawn at right angles. progeCAD helps you draw perfectly straight lines at exactly right angles with its *ortho* mode, short for "orthographic" — and another way of saying rectilinear.

Ortho mode is called a "toggle," because it only has two settings — on or off — just like a light switch. Because it is common to toggle modes while drafting, there are several ways to turn ortho mode on and off:

*   Pressing function key **F8**

*   Holding down the **Shift** key while drawing

*   Clicking **ORTHO** on the status bar

You can use these shortcuts to toggle ortho mode at any time, including during a command.

Selecting Entity Snap Modes

Entity snaps are among the most powerful tools in the progeCAD toolbox. "Esnaps" (for short) search for geometric features, such as the middle of a line or the center of a circle or the intersection of two entities. When the geometric feature is found, progeCAD snaps the cursor to that point and reports the name of the esnap it found, making for very accurate drawings.
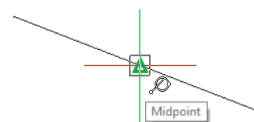
Like ortho mode, however, you sometimes want entity snaps on, and sometimes you want them off. You toggle esnaps during commands by using these shortcuts:

*   Press function key **F3**

*   Click **ESNAP** on the status bar

For turning on individual entity snap modes, I find the dialog box easier than the command line.

1. Enter the **ESnap** command, or else right-click ESNAP on the status bar, and then select **Settings**.

2. In the Settings dialog box, you can click **Clear** to turn off all esnap modes instantly.
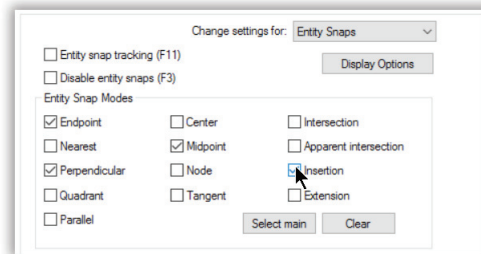
3. Now select the entity snaps that you think you will need when working with rectilinear drawings, like the apartment floor plan. Here are the ones I suggest you choose:

   ☑ **Endpoint** — snaps to the ends of lines, polylines, and arcs

   ☑ **Perpendicular** — snaps at right angles to lines and circles

   ☑ **Midpoint** — snaps to the midpoint of lines, polyline segments, and arcs

   ☑ **Intersection** — snaps to the intersection of two lines, circles, arcs, other entities, and combinations thereof

4. Click **OK** to exit the dialog box. The four esnap modes are now waiting and ready to spring into action when you use drawing and editing commands.

Calculating Scale Factors

One issue with CAD is that there are two kinds of *scale factors*. Normally, you draw everything "full size" in progeCAD. If a wall is 24 feet long, you draw it 24'-long in the drawing.

When it comes time to plot, the drawing must be scaled down to fit the paper. That 24-foot wall might be plotted just 8 inches long. The scale of the drawing fits 24' into 8", or 36:1. (That's 24' x 12 inches/foot = 8", or 288:8 = 36:1.) When plotted, everything in the drawing is 36 times smaller.

Just like in "Honey I Shrunk the Kids," there is a problem to making *everything* 36x smaller. There are some aspects of CAD drawings that cannot be many times smaller, such as text, linetypes, hatch patterns, dimension text, and arrowheads. These items should be "full size" when plotted. That means they must be 36x larger when drawn in CAD, in order to turn out 1x on the printed page.

Text, for instance, is typically 1/4" tall in paper drawings. That means it should be 9" tall in the CAD drawing. I arrived at 9" through 1/4" x 36.

To set the scale factor in progeCAD, use the commands and system variables for each type of scale-sensitive entity, as illustrated by the table below.

Entity	Command or System Variable
Linetype	**LtScale** sets the linetype scale factor.
Hatch Pattern	**HpScale** sets the hatch scale factor.
Dimension	**DimScale** sets the dimension scale factor.
Text	**Style** or **Text** set the height in plotted inches.

For this drawing, set the scale factors, as follows:

1. Use the **LtScale** system variable to preset the scale factor for linetypes:

    : **ltscale**
    Enter new value for LTSCALE <1.0000>: **36**

2. Use the **HpScale** system variable to preset the scale factor for dimensions:

    : **hpscale**
    Enter new value for HPSCALE <1.0000>: **36**

3. Use the **DimScale** system variable to set the scale factor for dimensions:

    : **dimscale**
    Enter new value for DIMSCALE <1.0000>: **36**

Text isn't scaled until later in this book when you learn use the Style and MText commands

Save As Template Drawing

By saving the current drawing as a *template*, you can later reuse it as the start for similar drawings. You can see the benefit to employing template drawings, where all settings are already established. You save yourself all the work you did in this chapter!

Templates are saved with the **SaveAs** command.

1. Enter the **Save As** command.
2. In the Save Drawing As dialog box, click the **Save As Type** droplist to change the extension of the file name from .dw*g* to .*dwt*.
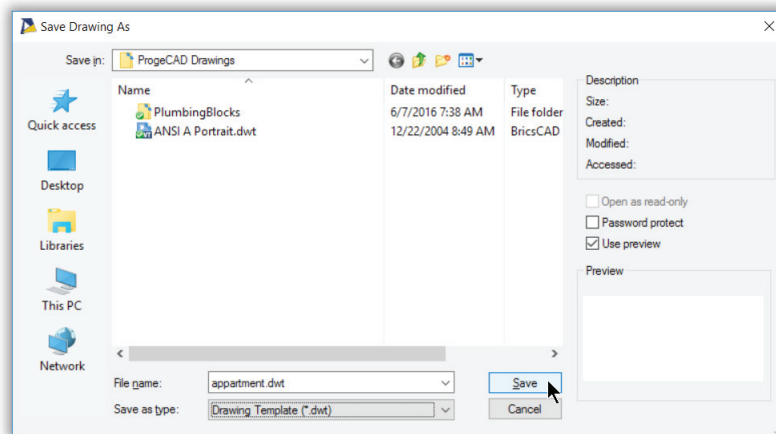
3. There is no need to change the file name. Click **Save**, and the drawing is saved as a template.

The next time, you can use the **Open** command's "Drawing Template (dwt)" file type to access it.

There is a problem, however. Notice the title bar reads "apartment.dwt". You need to work

with a drawing, not a template. (If you were to work with a template, all the lines you add in the following chapter are also saved to the template, and so would appear each time you start a new drawing with this template!)

4. Use the **SaveAs** command again, saving the drawing as a *.dwg* file with the name of "apartment.dwg."

———

Even though the drawing looks empty at the end of this chapter, it still contains information useful for the drawing exercises in the following chapters, such as the grid and snap spacing.

Drawing Exterior Walls

the *exterior walls* enclose the unit. Everything inside the apartment must fit between four walls. Two of the exterior walls (top and bottom) are 6" thick to provide extra insulation; the other two walls (left and side) are 4" thick, because they adjoin other units.

In this chapter, you draw the exterior walls with the **Line** and other commands (see figure below), and then save your work.
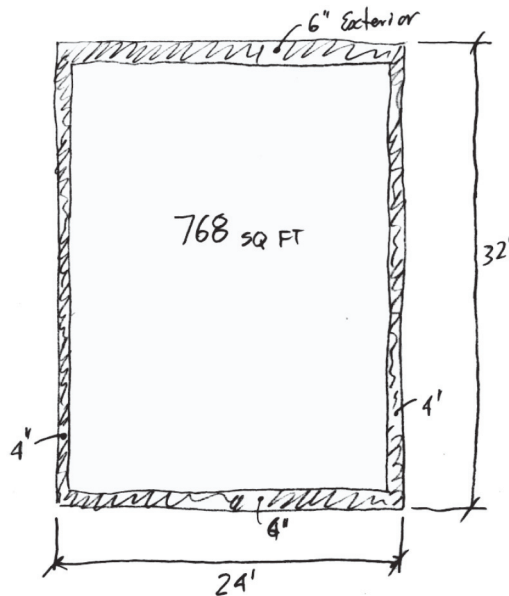
In This Chapter

- **Line** command
- **Offset** command
- **Trim** command

Drawing Lines

The **Line** command draws line segments. You use the command to draw the outline of the apartment's four exterior walls. You specify the position of lines by tying x, y coordinates (a.k.a. Cartesian coordinates); in later tutorials, you learn about other methods for inputting coordinates.

To draw lines with the **Line** command, you need to know the starting and ending x,y coordinate of the start (point 1) and end (point 2) of the line segment.

Point 1 ─┐
Point 2
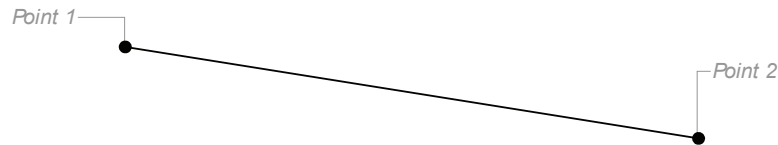
From the sketch of the exterior walls on the earlier page, we can see that two walls are 24' long, and that two are 32'. It is standard to begin drawing the apartment at the lower left corner of the drawing. This is known as the *origin*, where the x,y coordinates are 0,0. I've sketched the coordinates of each segment's endpoints in the figure below.

1. If necessary, use the **Open** command to open the *apartment.dwg* saved in the previous chapter.

2. Start the **Line** command.
Command: **line**

3. Specify where the first line segment should be drawn. Start the line at the origin (0,0):
Specify start point, ENTER to use last point or [Follow]: **0,0**

4. And then the line segment ends at 24' to the right — in the direction of x.
Next point or [Angle/Length]: **24',0**

The '0' part of the coordinate is the distance in the y direction, which is zero. Remember to include the apostrophe (') to tell progeCAD that the 24 is in feet; if you leave out the apostrophe, progeCAD assumes you mean inches, which is not what you want in this case.

5. The second segment is 32' tall, and so goes to x=24 and y=32:
Next point or [Angle/Length/Follow/Close/Undo]: **24',32'**

6. Draw the next segment is 24' to the left, and so goes back to x=0:
Next point or [Angle/Length/Follow/Close/Undo]: **0,32'**

7. To close the rectangle, you could enter the final coordinates (0,0), or you could use the **Close** option to have progeCAD draw the final segment for you.

    Next point or [Angle/Length/Follow/Close/Undo]: **c**

At this point, the drawing consists of a rectangle. If you do not see it all, use the **Zoom Extents** command.

```
0',32'                    24',32'



                          32 feet



    Y axis
    X axis

0,0 (the drawing origin)   24',0'
        24 feet
```
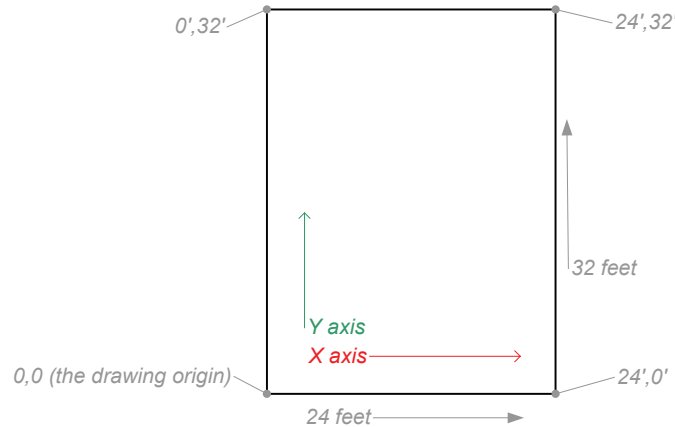
Offsetting Lines

The exterior walls are 4" and 6" thick. To show the thickness, you draw a second set of lines inside the large rectangle. You could use the Line command again, but there is an easier method. The **Offset** command makes copies of entities by drawing parallel duplicates that are offset by a distance that you specify.

To use the Offset command, you need to know the following items:

- The entities to be copied
- The offset distance
- The side(s) on which to draw the parallel copy

Start by creating the 4"-thick inside walls:

1. Start the **Offset** command.
    Command: **offset**

2. Specify the offset distance of 4 inches:
    Specify offset distance or ENTER for Through point: **4**

3. Select the entity to copy.
    Select entity or [Undo]: *(Pick one of the vertical lines)*

OFFSET COMMAND

Makes parallel copies.
Command: **offset**
Aliases: **o**, **parallel**
Toolbar: **Modify | Offset**
Menu: **Modify | Offset**

ALTERNATIVES

Copy makes copies.
Mirror makes mirrored copies.
Array makes multiple copies.

4. Pick a point inside the rectangle. progeCAD will make the copy through that point.

Specify side for parallel copy or [Both sides]: *(Pick inside the rectangle)*

Step 3: Pick the vertical line

Step 4: Pick a point inside the rectangle

Step 5: Repeat on this side
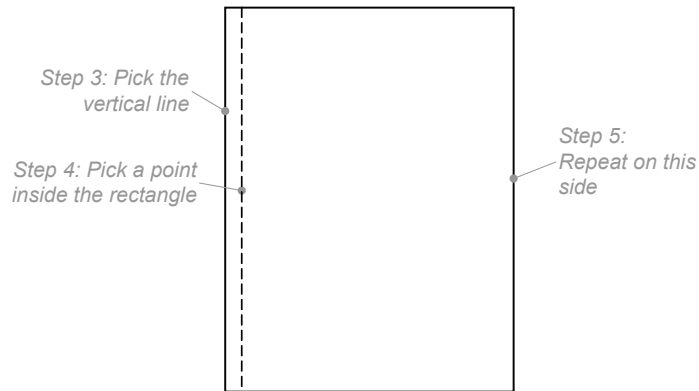
5. The command repeats so that you can pick the other vertical line:

Select entity or [Undo]: *(Pick other vertical line)*

Specify side for parallel copy or [Both sides]: *(Pick inside the rectangle)*

6. Press **Enter** to exit the command.

Select entity or [Undo]: *(Press **Enter**)*

TIP   Some command repeat themselves, like **Offset**. This allows you to repeat the editing or drawing operation without having to restart the command.
         To end a repeating command, press **Enter** or **Esc** on the keyboard. As an alternative, you can select **Cancel** from the prompt menu, or right-click until the command ends.

At this point, your drawing consists of a rectangle with two extra vertical lines, as shown below.
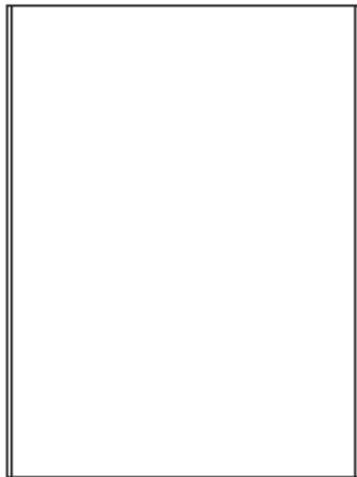
Repeat the **Offset** command to add the 6" offset lines inside the top and bottom of the rectangle.

Command: **offset**

Specify offset distance or ENTER for Through point<0'-4">: **6**

Select entity or [Undo]: *(Pick one of the horizontal lines)*

Specify side for parallel copy or [Both sides]: *(Pick inside the rectangle)*

Select entity or [Undo]: (Pick the other horizontal line)

Specify side for parallel copy or [Both sides]: *(Pick inside the rectangle)*

Select entity or [Undo]: *(Press **Enter** to end the command)*

The result should look like the figure illustrated below.
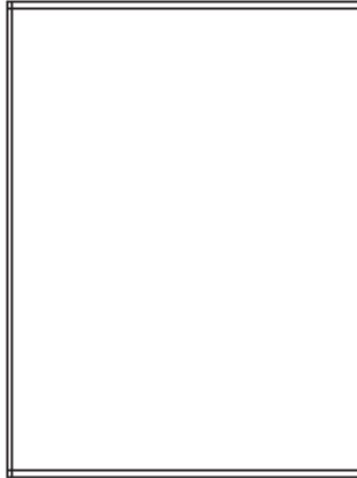
Trimming Lines

Creating the parallel copies resulted in messy intersections where the lines overlap. You use the **Trim** command to clean up intersections.

Trim works by shortening entities back to a *cutting* entity. Because the four inner lines overlap each other, you can use them as cutting entities *and* the entities to be trimmed simultaneously.

To use the **Trim** command, you need to know:

• The cutting entities

• The entities to be trimmed

Line to be trimmed back   *Line used as the cutting edge*   *Portion of line to be trimmed back*
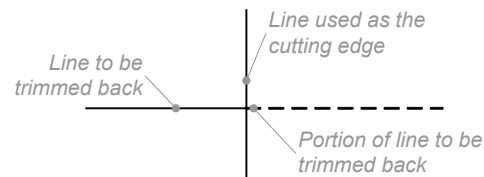
1. Start the **Trim** command.
Command: **trim**

2. Select all the entities in the drawing by pressing **Enter**.
Select cutting entities for trim or ENTER to select all: *(Press **Enter**)*
Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]:

Portion of line to be trimmed back
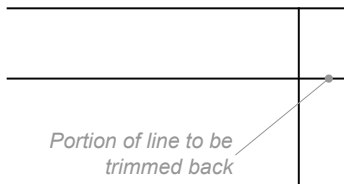
3. Pick a line to trim. This can be tricky: *pick the portion of the line you want removed*; do not pick the portion you want to retain.

    Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Pick the end of a line)*

    Line trimmed
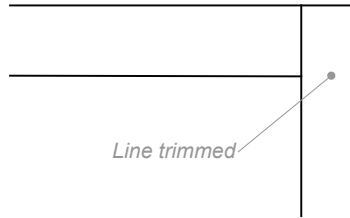
4. Repeat seven more times to trim the ends of all four inner lines.

    Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Pick another segment)*

5. When done, press **Enter** to end the command.

    Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Press **Enter**)*

At this point, your drawing should look like the double-line rectangle below.
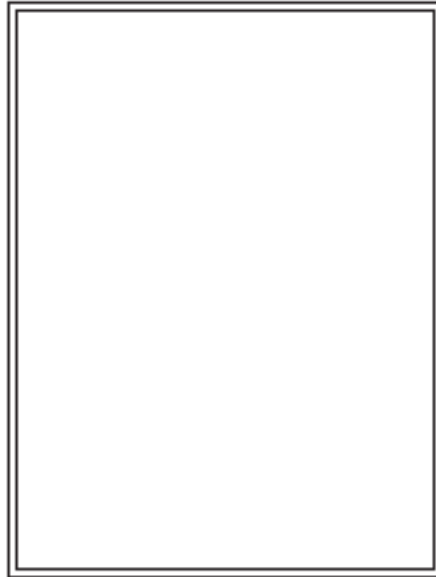
QSAVE COMMAND

Saves drawings.

Command: **qsave**

Menu: **File | Save**

Toolbar: **Standard | Save**

Shortcut: **Ctrl+S**

ALTERNATIVES

Save prompts for file name.

Exports saves in other formats.

Be sure so save your drawing! Use the **QSave** command, or press **Ctrl+S**.

Defining Interior Spaces

tHe interior spaces contain the living room, kitchen, a single bedroom, the closets, and so on. They are constrained by the exterior walls, and each other: give one area more space and it takes away space from other areas. In general, the kitchen and bathroom receive less space, while major living areas — dining, living, and bedroom — receive more space.
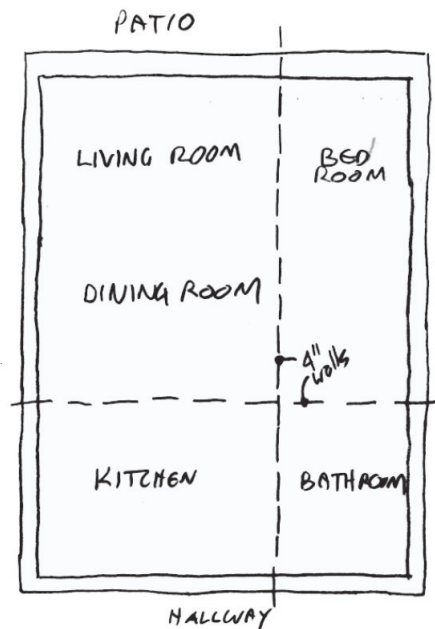
For this project, each apartment unit is subdivided into four major areas:

In This Chapter

- **ExpLayers** command
- **Move** command
- **Erase** command
- **Lengthen** command
- **Redraw** command
- **Fillet** command

- Living and dining room
- Kitchen (kitchen shares space with the entrance hall)
- Bathroom (bathroom shares space with coat closet)
- Bedroom, which has three smaller areas:

        Bedroom closet

        Ensuite bathroom (sink and toilet only; no bathtub or shower)

        Hot water tank closet

All interior walls are 4" thick.

Creating Layers

When drawings consist of more than just a few lines, it becomes important to segregate them. For example, external walls are built differently from internal walls, and so they should be segregated. Similarly, the content of rooms (furniture, appliances, and so on) should be segregated from the content of walls (doors, window, and such).

There are two ways to segregate information in CAD drawings: through *layers* (discussed in this chapter) or through *externally-referenced drawings*. Externally-referenced drawings keep entire drawings separate from each other, yet display as a single drawing. Here we concern ourselves with layers only.

LAYER COMMAND

Creates and controls layers.

Command: **layer**

Aliases: **la, ddlmodes**

AutoCAD: **layer**

Menu: **Settings | Explore Layers**

Toolbar: **Entity Properties | Explore Layers**

ALTERNATIVES

ExpLayer makes and controls layers at the Explorer.

SetLayer sets current layer from selected entity.

About Layers

Layers are like transparent sheets of paper; on each sheet you draw different parts of a drawing, such as the walls, the electrical, and the furnishings. To see the entire drawing, we overlay all the sheets. An electrician, however, is interested in only the walls and the electrical; he doesn't care about the furnishings. To generate the electrical plan, we would turn off the furnishings layer.

Entities are separated from each other by assigning them to layers. (If you don't do this, then all entities end up on a single layer, which happens to be named "0.")

Layers have properties that determine the appearance of entities. The properties include color, line type, visibility, and others. This aspect of layers is powerful, because it means that all entities of a layer can have their properties changed at once:

- Change a layer's color from black to red: all entities assigned to that layer also change from black to red
- Change the layer's line type from Continuous to Hidden: the entities' line type change to Hidden

- Turn off a layer, and its entities disappear from the screen; turn on the layer, and the entities reappear

There is no limit to the number of entities that can be assigned to a layer. There is no limit to the number of layers that can be created in a drawing. But in practice we use just a few layers, because dealing with hundreds or thousands of layers gets annoying. Well, huge design projects like skyscrapers use thousands of layers to the benefit of all the disciplines involved and the design stages, like proposed, to be changed, under construction, as built, and to be decommissioned.

Layer Standards

It is common for architectural and engineering firms to establish *layer standards*. These specify the names and properties of layers, and other aspects of drawings. Standards are usually specific to disciplines because the layer names used for architectural drawings are not suitable for mechanical drawings.

Standards are sometimes dictated by clients or by industry associations. Examples include the US Coast Guard and the Architectural Institute of America. See http://www.usbr.gov/foia/Vol%20III/NCS-CAD_Layer_Guidelines.pdf.

TIPS It is good practice to create a template drawing that containing all of the layer names used by your firm.

You can copy layers from one drawing to another: use the Explorer window's **Copy** and **Paste** commands.

Properties of Layers

Layers have the following properties:

Status (Current) — only one layer can be current at a time. It is known as the *working* layer; when new entities are drawn, they get assigned to the current layer automatically. If an entity ends up on the wrong layer, don't worry: it can be moved with the LayMch command.

Name — any combination of letters, numbers, and spaces up to 255 characters long. The name cannot contain thesymbols / ? : > " < | and \ . You can use descriptive names ('Walls', 'Electrical', 'Furnishings') and prescriptive names ('A-WALLS', 'E-LIGHTS').

On or **Off** — layers can be on (visible and editable) or off (not visible). The default is on. On/Off is rarely used, as Freeze/Thaw is more efficient.

Freeze or **Thaw** — layers can be frozen (not visible and ignored) or thawed (visible). The default is thawed.

Lock or **Unlock** — layers can be locked (visible but not editable) or unlocked (visible and editable). The default is unlocked.

Color — layers can be assigned any of progeCAD's colors. The default color is black.

Linetype — layers can be assigned any linetype loaded into the drawing. The default linetype is 'Continuous'.

Lineweight — layers can be assigned any of progeCAD's 24 line weights, from 0 mm to 2.11mm wide. The default weight is 'Default', and the default Default is 0.25mm. You can changed the default through the **Settings | Display | Lineweights**.

Transparency — layers can be dimmed by making them more transparent, ranging from 0 (fully opaque) to 90 (nearly transparent). The default is 0.

New VP Freeze — specifies that when a new viewport is created that the layer be frozen. Default is not frozen.

Plot — layers can be plotted or not plotted. The default is to be plotted.

(Plot Styles names are specified only when drawings use named plot styles; otherwise, the plot style column is grayed out.)

Frozen Layers

The difference between *off* and *freeze* is subtle. When a layer is turned off, its entities are no longer visible, but they are taken into account when the extents of the drawing are calculated, such as during the **Regen**, **Zoom**, and **Plot** commands. As well, you can continue to draw on layers turned off, but the entities remain invisible until the layers are turned on.

When a layer is frozen, the extents of its entities are ignored, and you cannot draw on it.

In addition, frozen layers occur in two different ways: in *current* viewports and in and *new* ones. (These are the viewports found in paperspace or layout mode.) By freezing layers in individual viewports, you create different kinds of views of drawings. The "New Viewports" option determines whether layers are frozen when new viewports are created.

ByLayer and ByBlock

You may have seen colors or line types called "BYLAYER." This special name means that entities take on the color, line type, and other properties specified by their layer — sort of like an automatic color and line type.

Similarly, "BYBLOCK" colors and linetypes mean that entities take on the color and other properties specified by the block of which it is a part.

Local Overrides

I have described the preferred method of controlling the properties of entities: by layers. This can be overridden, a practice that CAD experts frown upon. For instance, you could assign the color green to an entity whose layer color is red. This is called a "local override."

Local overrides can be applied with the **Color** and **Linetype** commands, or with the **Entity Properties** toolbar: select the entity, and then change its color or line type.

Exploring the Explorer

progeCAD's Explorer provides a detailed view into the elements that make up drawings — just about everything, except for entities and system variables. Explorer lists the names of open drawings, the states of layers, line types, text styles, UCSs (named coordinate systems), named views, layouts, user-defined blocks, dimension styles, groups, and external references. (Auto-CAD's DesignCenter is similar to the Explorer.)

Explorer is used to inspect the status of elements, to create new ones, and to change their properties. Perhaps the most powerful feature of Explorer is the ability to copy elements among drawings.
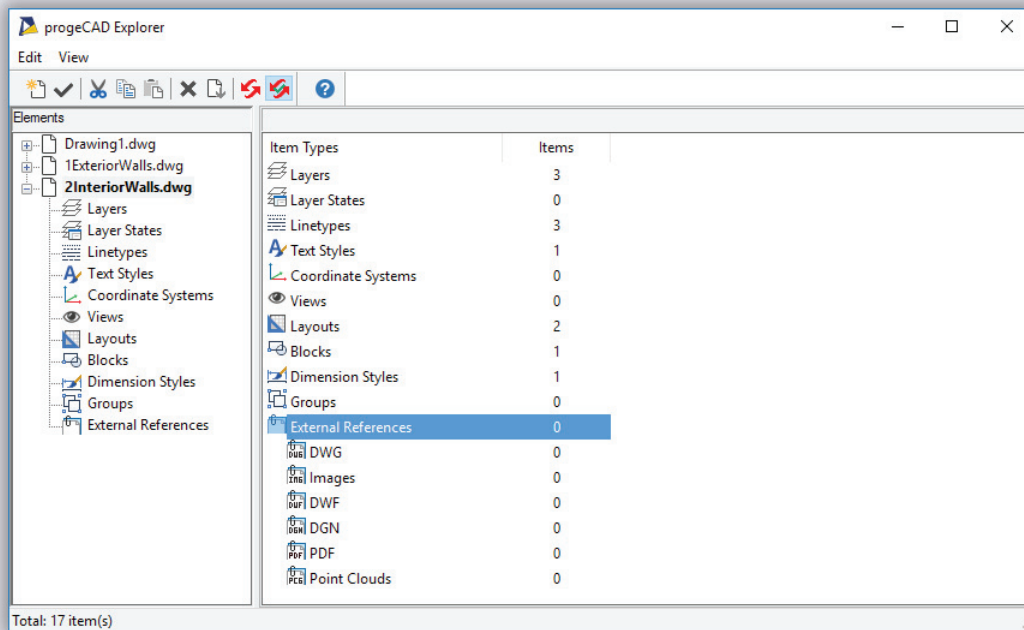
The window is opened by the **Explorer** command. Related commands open the window with the specified element showing:

      ExpBlocks — blocks stored in drawings

      ExpDimstyles — dimension styles defined in drawings

      ExpFonts — text styles names

      ExpGroups — group names

      ExpLayers — layers and properties

      ExpLayerStates — layer states

      ExpLayouts - layout names and properties

      ExpLtypes — line types loaded into drawings

      ExpUcs — user-defined coordinate systems

      ExpViews — named views stored in drawings

Toolbars

You can manipulate elements in a variety of ways. One way is through the toolbar:

New Item — creates a new element with a generic name and default properties.

Current — makes the selected element current, such as the working layer or view.

Cut — copies selected elements to the Clipboard, and then erases them from the drawing.

Copy — copies selected elements to the Clipboard.

Paste — pastes elements from the Clipboard into the drawing, and adds them to the Explorer.

Delete — removes the element from the drawing; certain elements cannot be deleted and so progeCAD tells you that via this dialog box:
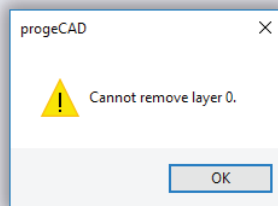
Purge — removes unused elements, such as empty layers and unused text styles.

Regen — forces the drawing to update to reflect changes made in the Explorer window.

On/Off Regen — toggles automatic regeneration.

Some elements display extra buttons on the toolbar. Layer elements:

New Property Filter — creates a new filter based on layer properties

New Group Filter — creates a new filter based on a group of layer names

Invert Filter — inverts the layer names displayed by the filter

Layer States Manager — displays the layer states manager dialog box

Export Layer Filter — exports the filter to an external file for use in other CAD systems

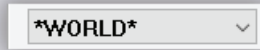**Filter** — determines which layers are displayed; * = all.

Layer States elements:

Edit Layer State — edits the properties of the selected layer state

Import Layer State — imports the layer state settings from an external LAS file

Export Layer State — exports the layer state settings to a LAS file

Coordinate Systems elements:

World — returns a UCS to the WCS view

View Manager elements:

View Manager — displays the View Manager dialog box

Layouts elements:

Icons — displays layouts in Explorer as icons

Details — displays layouts with properties

Template — saves the current drawing as a DWT template file

Save — saves the current drawing as a DWG file

Block elements:

Icons — shows blocks as preview images.

Details — lists blocks by name and details, such as number of insertions and insertion points.

Insert — inserts the selected block in the drawing.

Insert external file — displays Insert Block dialog box to insert any drawing as a block.

Save blocks — runs the **WBlock** command to export blocks as *.dwg* files.

Edit Block Reference — runs the **RefEdit** command to edit blocks.

Group elements

Highlight — highlights the entities that are part of the selected group

Add Entities — adds entities to the current group

Remove Entities — removes entities from the current group

Reorder Entities — changes the order that entities appear in the group

External References elements:

Icons — shows xrefs (external drawings attached as references) as preview images.

Details — lists xregs by name and details

Insert — inserts the selected block in the drawing.

Attach — attaches a DWG file as a reference to the drawing

Xref Manager — displays the Xref Manager dialog box for managing xrefs

Image Manager — displays the Image Manager dialog box for managing images

Shortcut Keystrokes

Within the Explorer window, you can use the following shortcut keystrokes:

Ctrl+A — selects all elements

Ctrl+C — copies selected elements

Ctrl+P — purge elements

Ctrl+V — pastes elements

Ctrl+X — cuts selected elements

Del — deletes selected elements
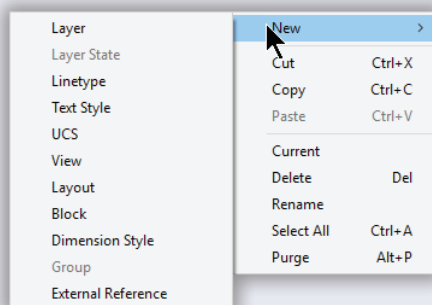
F1 — displays help window

Mouse Actions

To select an element, click on it.

To select a range of elements, hold down the **Shift** key, and then select the first and last elements in the range.

To select random elements, hold down the **Ctrl** key, and then select each element.

You can right-click elements to display a shortcut menu of options.

Layer Command

You create layers with the **Layer** commands. (Alternatively, you can also use the ExpLayers command, but Layer is more like AutocAD's.)

If you had problems completing the previous tutorial, the exterior walls drawing is available as *1ExteriorWalls.dwg*.

In this tutorial, you create layers to segregate details. As I noted earlier, layers can have *any* name — so what names should you use? Spend a moment thinking about the content of the apartment design. It consists of these elements:

- Walls
- Doors and windows
- Fixed assets
- Non-fixed assets

A "real" drawing would also have plumbing, electrical and communications, alarm system, heating and ventilating, and many other details — and so many more layers.

For this tutorial, use the above phrases as the layer names. To create layers, follow these steps:

1. Start the **Layer** command. Notice the Layer Properties Manager palette, and that it already has one layer named '0'. This is the default layer and is present in every new drawing. In addition, layer 0 has some special properties, and so cannot be erased.
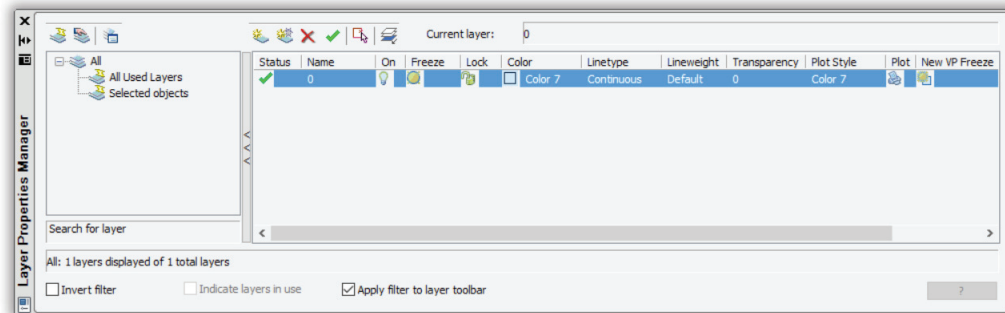
(A palette is different from a dialog box: it can always stay open, while a dialog box must be closed before you can continue drawing and editing.)

2. In the toolbar, click the **New Layer** button. Notice that progeCAD creates a new layer with the same properties as the selected layer: white color, continuous linetype, and so on.
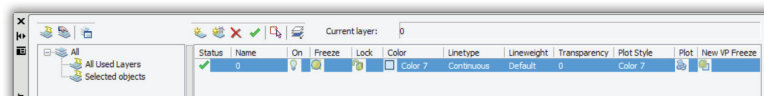
3. progeCAD names the new layer "Layer1." Change its name to "Walls" by clicking the name and then changing it.

4. Add the remaining layers by clicking New and then giving them the following names:

> Doors and Windows
> Assets-Fixed
> Assets-Nonfixed

Here's the reason for using names with reversed words — such as "Assets - Fixed", instead of "Fixed Assets." progeCAD lists layer names in alphabetical order, and not in the order in which layers are created. By reversing the parts of the names, all Asset layers are listed together.

5. Now select **Walls**, and then click the **Current** ✔ button. This makes the layer current (i.e., the working layer), because the next lines you draw will be the interior walls. When done, the dialog box should look like the figure above.

6. Close the palette by clicking the **X** button.

7. On the Entity Properties toolbar, notice that the Layer Control displays "Walls."
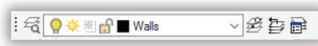
Above: Entity Properties toolbar; ***below***: Layers panel of the ribbon
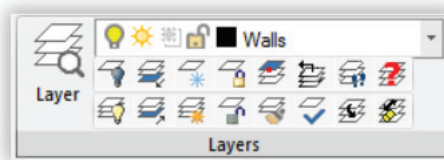
If progeCAD is running the Ribbon Menu workspace, then the Home tab of the ribbon, the Layers panel reports "Walls" as the current layer.

Assigning Entities to Different Layers

Currently, all the lines, which represent exterior walls, are on layer 0. They really should be on layer "Walls." I'm going to show you a quick way to move entities to another layer.

1. Press **Ctrl+A** to select all entities in the drawing.
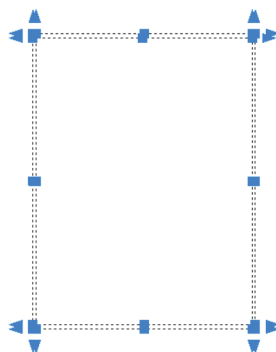
Notice that the lines are highlighted by dashed lines and blue *grips*. Grips allow you to edit entities directly, and are discussed later.

2. From the Layer Control on the toolbar or the ribbon, select "Walls."
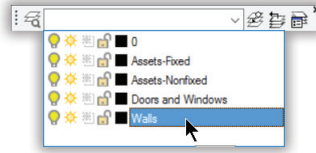
And you're done. Ja, it's that easy.

Adding Interior Lines

With the layers set up, you can now add the interior walls. The technique is the same as for the exterior walls:

a. Draw one side of the wall with the **Line** command

b. Add the other side of the 4"-wall with the **Offset** command

c. Clean up overlapping lines at intersections with the **Trim** command
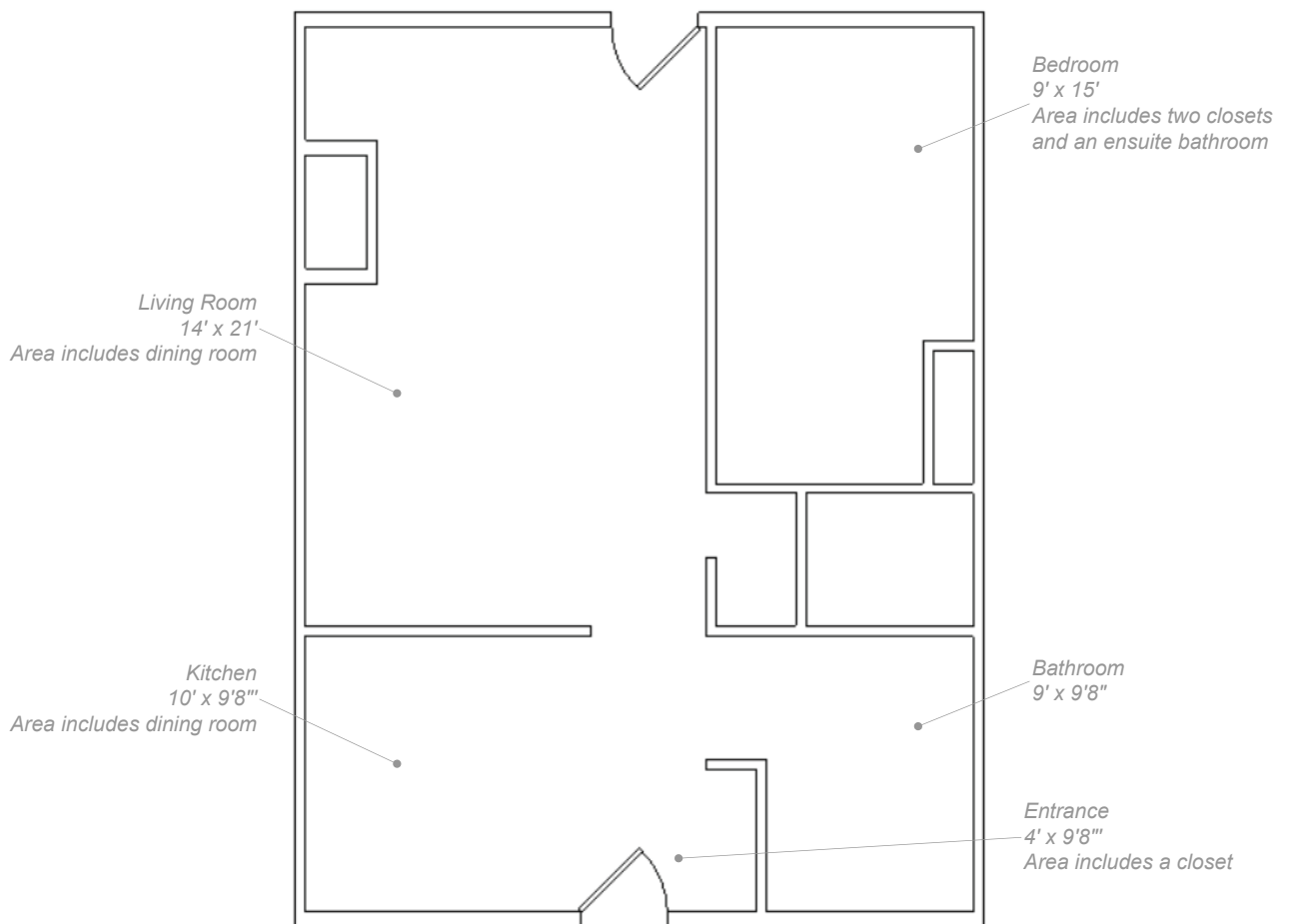
You can make the rooms as big or little as you like, as long as they fit the exterior walls. Obviously, some rooms must be bigger than others, as indicated by the dimensions illustrated below (width x depth).

Here are some design hints:

- Interior walls are 4" thick
- You don't need to create openings for doors or windows, because those will be indicated later when you insert door and window symbols
- The fireplace is roughly 4' x 2'

When you add up dimensions, don't forget to allow for the wall thicknesses; otherwise, you end up with incorrect totals. Across the top of the apartment, for example, the living room + bedroom are 14' + 9' = 23' wide. Yet the apartment unit is 24' wide. Where did the missing foot go? The answer is in the thickness of the walls: the two outside walls and one inner wall are each 4" thick. That's 3 x 4" = 12" = 1'.

Starting Offset Lines

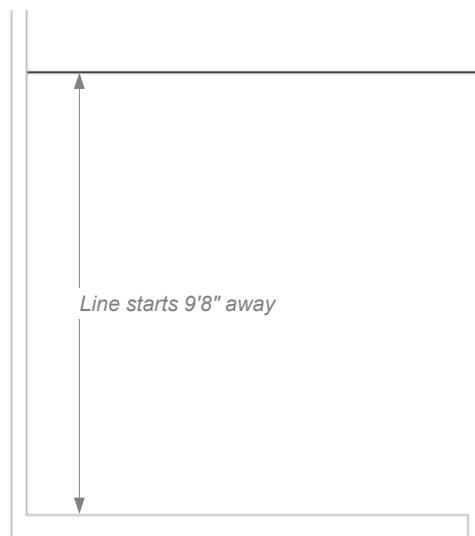Drawing lines that start at the end of existing lines is pretty easy: use the ENDpoint entity snap. But how do you draw a line that starts 9'8" from the end of another line, such as the wall that separates the kitchen from the dining/living room?

Line starts 9'8" away

One way is to use the Offset command, but I'll show you several other ways to do it so that you get a feel for the flexibility progeCAD provides in approaching drafting tasks:

- Draw a temporary line, then erase it later
- Draw construction lines
- Calculate the coordinates
- Draw, then move the line
- Offset the line, then lengthen it to size

Try each one to see which works best for you. You will find that in some cases one approach works better than another.

Draw, Then Erase Temporary Lines

The brute force method draws a temporary line to specify the distance, draws the wanted line, and then erases the temporary one. See figure below. I like this approach, because it is clear to me what is happening.

<table>
<tr><td>

ERASE COMMAND

Deletes selected entities.

Command: **erase**

Aliases: **e, delete**

Keyboard: **Del**

Menu: **Edit| Delete**

ALTERNATIVES

CutClip copies selected entities to the Clipboard, and then erases them from the drawing.

Oops returns the last erased entity(ies).

U undoes this command.

</td><td>

1.   This approach requires that ortho mode be turned on. If necessary, click the ortho button on the status bar so that the icon is shown in blue and/or the word in black, not gray.

2.   Use the **Line** command to draw the temporary line, like this:

Command: **line**

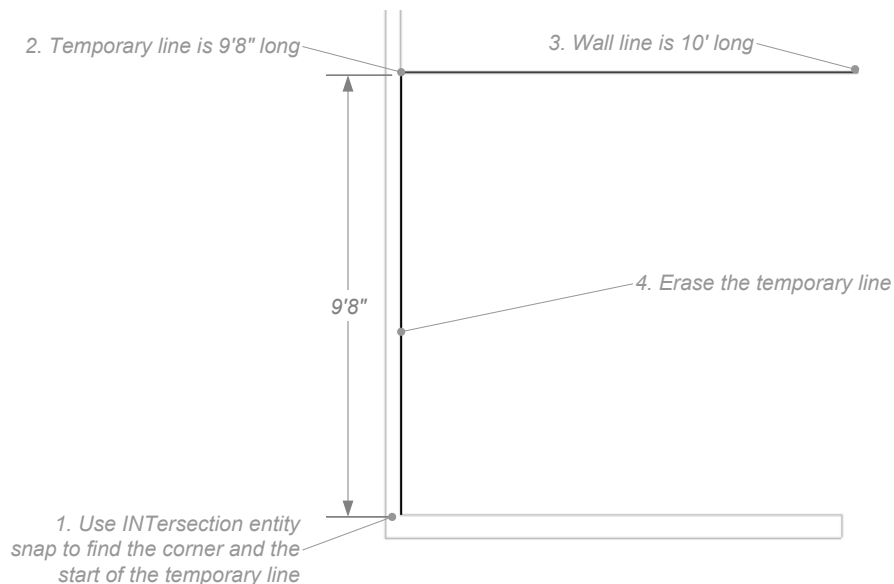Specify start point, ENTER to use last point or [Follow]: **int**

Snap to intersection of: *(Pick intersection of the lines at point #1, as shown above)*

Next point or [Angle/Length]: *(Move cursor upwards in the y direction, point #2)* **9'8**

3.   Continuing with the Line command, draw a second line 10 feet long at a right angle:

Next point or [Angle/Length/Follow/Undo]: *(Move cursor to the right, point #3)* **10'**

Next point or [Angle/Length/Follow/Close/Undo]: *(Press **Enter** to exit the command)*

</td></tr>
</table>

2. Temporary line is 9'8" long

3. Wall line is 10' long

4. Erase the temporary line

9'8"

1. Use INTersection entity snap to find the corner and the start of the temporary line

---

TIP   The **Line** command doesn't always require a pair of coordinates. Above, you entered a single value, 9'8, for the coordinate. How does progeCAD know if you mean x or y? When you turned on ortho mode, progeCAD restricted cursor movement in the x (horizontal) and y (vertical) directions.

When you move the mouse (cursor) up, progeCAD understood you meant the y direction. And so when you entered the distance of 9'8, progeCAD understood that to mean y=9'8.

---

4. Use the **Erase** command to remove the temporary line, as follows:

Command: **erase**

Select entities to delete or [FILter/Qselect]: *(Pick the temporary line, point #4)*

Entities in set: 1

Select entities to delete or [FILter/Qselect]: *(Press **Enter** to exit the command)*

5. The line is erased. If necessary, use the **Redraw** command to clean up the display.

Command: **r**

Construction Line Method

The *construction line* method creates a layer for construction lines, and then uses the **InfLine** command to place a grid of construction lines. Following that, you switch back to the interior-walls layer, and trace over the construction lines with the **Line** command.

Calculated Coordinates Method

The *calculated coordinates* method works out the x,y-coordinate of the line's start point.
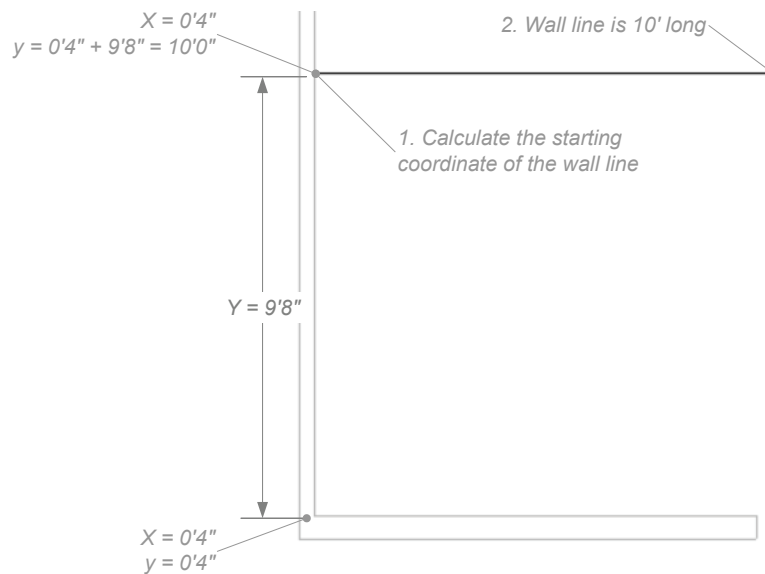
The figure above illustrates the math. The x coordinate stays the same, but the y coordinate changes, as follows:

$$y = 0'\text{-}4" + 9'8"$$
$$= 10'0"$$

(Having to add up feet and inches just about makes you pine for the metric system, eh?) You now draw the line using the coordinates you worked out:

Command: **line**

Next point or [Angle/Length]: **4,10'**

Next point or [Angle/Length/Follow/Undo]: *(Move cursor to the right in the x direction)* **10'**

Next point or [Angle/Length/Follow/Close/Undo]: *(Press **Enter** to exit the command)*

Moved Entity Method

The *moved entity* method draws a line the correct length, and then uses the **Move** command to move it 9'8" away.

1. With the **Line** command, draw a line that starts at 0,0 (point #1) and is 10' long, as illustrated above.

2. Start the **Move** command, and then select the line you drew with the Last (l) selection option:

Command: **m**

Select entities to move or [FILter/Qselect]: **l**

Entities in set: 1

Specify base point or [Displacement] <Displacement>: *(Press **Enter** to end entity selection)*

TIP Notice the use of "l" in the 'Select entities to move' prompt, above. It is short for "last" and tells progeCAD to select the last-created (or most-recently created) entity in the drawing.

There are numerous selection options, but here are the ones that I use the most:

a	Selects all entities in the drawing
c	Prompts to create a selection rectangle that includes all entities crossing and within the rectangle
l	Selects the last-drawn entity
p	Selects the previously-selected entities
w	Prompts to create a selection rectangle that includes all entities within the rectangle
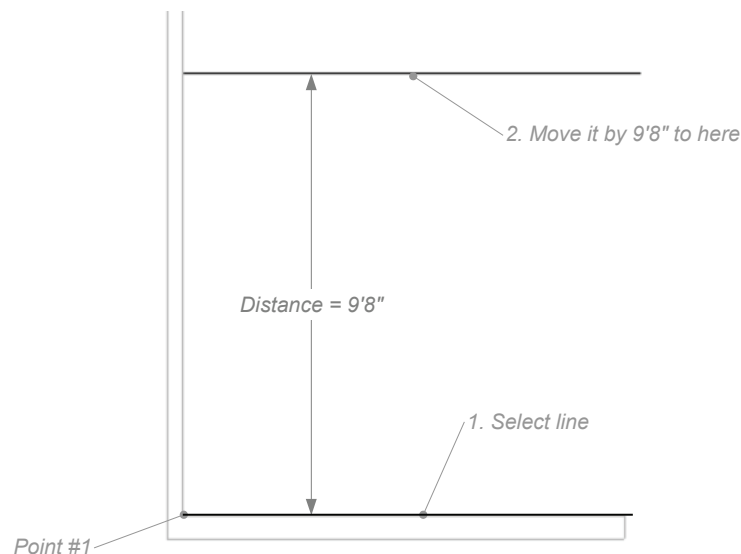
---

2. Move it by 9'8" to here

Distance = 9'8"

1. Select line

Point #1

3.   Now specify a distance of 9'8" to move the line. To be sure the action is accurate, used ENDpoint entity snap to find the start point of the move, as follows:

Vector/<Base point>: **end**

Snap to endpoint of: *(Pick end of line at 1; see figure below)*

Displacement point: **0,9'8**

Offset-Lengthen Method

The offset method uses the **Offset** command to offset an existing line, and then uses the **Lengthen** command to correct its length.
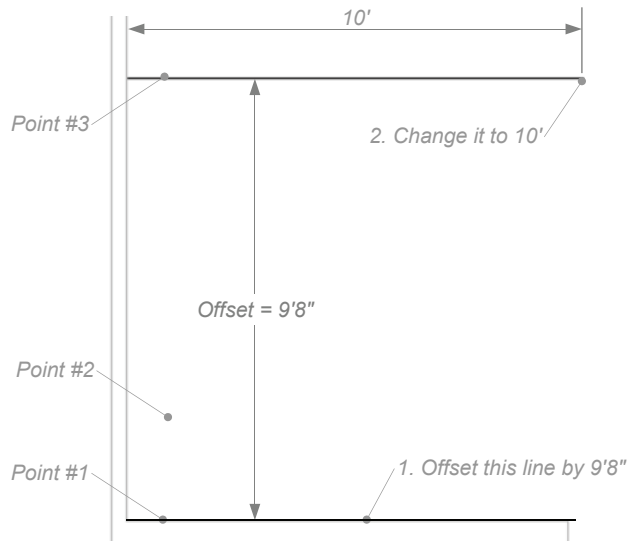
1.   Use the **Offset** command to create the offset copy, as follows:

    Command: **offset**
    Offset: Through point/<Distance> <0'-1">: **9'8**
    Select entity: *(Pick line at point #1)*
    Both sides/<Side for parallel copy>: *(Pick above line at point #2)*
    Select entity: *(Press **Enter** to exit the command)*

2.   The line that was offset is the wrong length. Use **Lengthen** command to change it to ten feet, as follows:

    Command: **lengthen**
    Edit length: Select entity to list length or [DYnamic/Increment/Percent/Total]: **t**
    Enter total length or [Angle] <0'-0">: **10'**
    Select entity to change or [Mode]: *(Pick line at point #3)*
    Select entity to change or [Mode/Undo]: *(Press **Enter** to exit the command)*

The "t" option specifies the *total*, new length of the line. The command's options include:

Lengthen	Meaning
DYnamic	Interactively changes the length as you move the cursor
Increment	Adds to the entity's length by this amount; a negative value subtracts from the length
Percent	Changes length by a percentage:
	Less than 100 = entity is shorter
	100 = no change
	More than 100 = entity is longer
Total	Specifies the absolute length, longer or shorter

TIPS The **Lengthen** command applies only to open entities, such as lines, arcs, and polylines.

Your pick point at the 'Select entity to change' prompt determines from which end the entity is lengthened or shortened. The endpoint closest to the pick point is the end that changes; the other end point remains in place.

Alternate Strategy for Trimming Lines

The different approaches described above show the flexibility of progeCAD; it usually provides more than one way to perform operations. Earlier, I suggested you draw the interior walls the same way as the exterior walls: place lines, make offsets, and trim the excess. Here is an alternative method to trimming.

FILLET COMMAND

Rounds off intersections.
Command: **fillet**
Aliases: **f**
Menu: **Modify | Fillet**
Toolbar: **Modify | Fillet**

ALTERNATIVES

Chamfer angles off intersections.

Trim shortens open entities to a boundary.

The obvious way to trim intersections is with the **Trim** command to cut off unwanted portions of lines. Less obvious is the **Fillet** command; in fact, Fillet is preferred over Trim by most CAD operators.
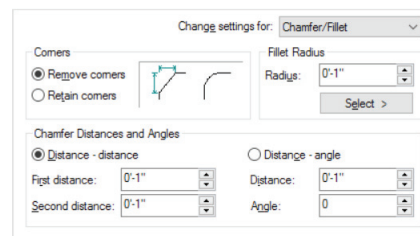
1.   Start the **Fillet** command, and then enter "s" for the **Settings** option, as follows:

Command: **fillet**
Current settings: mode = TRIM, radius = 0'--1"
Fillet: Select first entity or [Polyline/Radius/Trim/Settings/Multiple]: **s**

2.   progeCAD displays the Drawing Settings dialog box opened to the Entity Modification | Chamfer/Fillet tab. Here you set the fillet radius.

To trim lines with fillets:

    a.   Set the radius to 0" in the Fillet Radius area by changing **Radius** to **0**.

    b.   Leave the **Corners** option set to "Remove Corners."

    c.   Click **OK** to exit the dialog box.

3. progeCAD returns to prompting you in the command bar. Notice that the radius is now set to 0". Pick the first line:

    Fillet: Select first entity or [Polyline/Radius/Trim/Settings/Multiple]: *(Select a line)*

Important: Pick the part of the line you wish to keep, as illustrated below.

Portions of lines to be retained

Fillet creates clean intersection
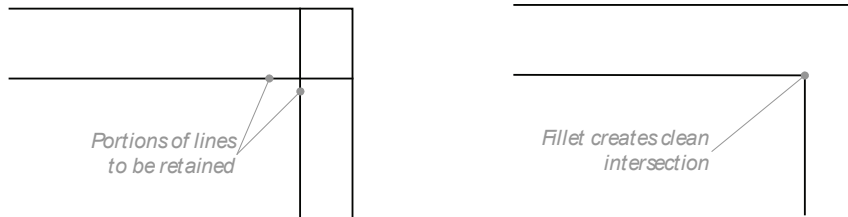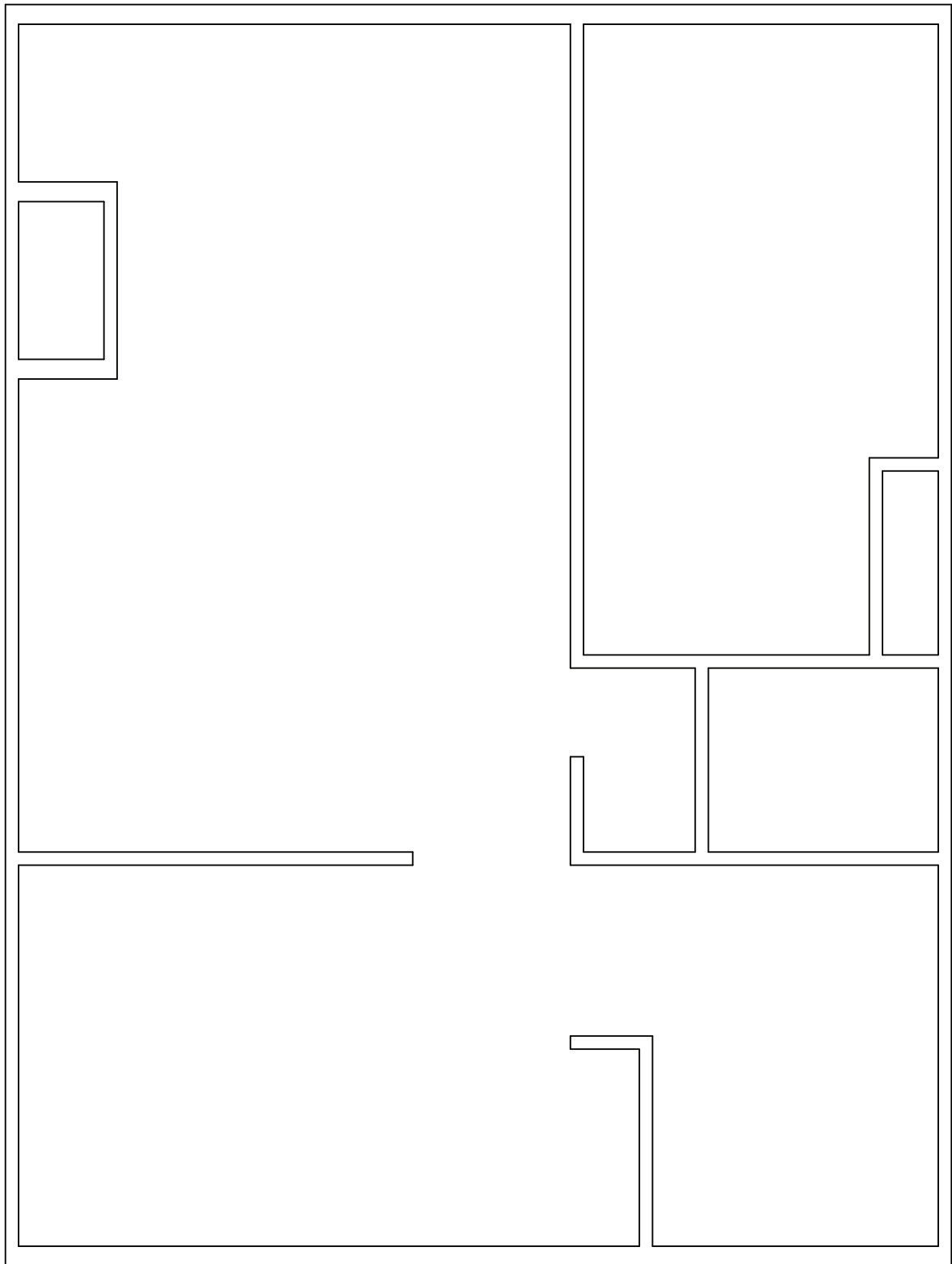
4. And then select the second line:

    Select second entity: *(Select an intersecting line.)*

progeCAD trims the intersection by cutting back both lines to where they intersect.

———

At this point, your drawing should look similar to the floor plan shown below.

 (If you had problems completing the previous tutorial, the apartment drawing is available as *2InteriorWalls.dwg*.)

Printing Drawings

at this point, your drawing should be looking good on your screen, but it would be great to see it on paper. Let's print the drawing now. Drawings are plotted with the **Print** or **QPrint** commands.

(If you had problems completing the previous tutorial, the apartment drawing is available as *2InteriorWalls.dwg*.)

Plotting vs. Printing

In the past, plotting and printing had different meanings:

*   *Plotting* meant sending drawings to vector devices such as pen plotters. Twenty to thirty years ago, plotters produced much higher quality output than did printers, because these raster devices created cruder plots than did vector devices.

*   *Printing* used to mean outputting drawings to raster devices such as dot-matrix printers. In the last ten years, raster technology has been refined to the point that nearly all CAD drawings are output on raster devices — either ink jet printers or laser printers. The quality is as good as the pen plotters of yore, but the speed is much faster.

In This Chapter

*   **QPrint** command
*   **Preview** command

*   **Print** command
*   **Export** command

*   Headers and footers

More common than printing today is *electronic plotting*, where drawings are saved in file formats suitable for sending by email or displaying at Web sites. The most common "eplot" formats are Acrobat PDF and DWF, both of which can be created by progeCAD.

Because they now mean the same thing, I use the words "plot" and "print" interchangeably In this chapter.

Plotting, Basically (Print)

To control the plotting process, you use the **Print** command. It displays the Print dialog box with its many options. For most plots, you do not need to worry about most options, and so this part of the chapter describes the basic options to use. (A complete description of the Print dialog box comes later in this chapter under the "Advanced Options" heading.)
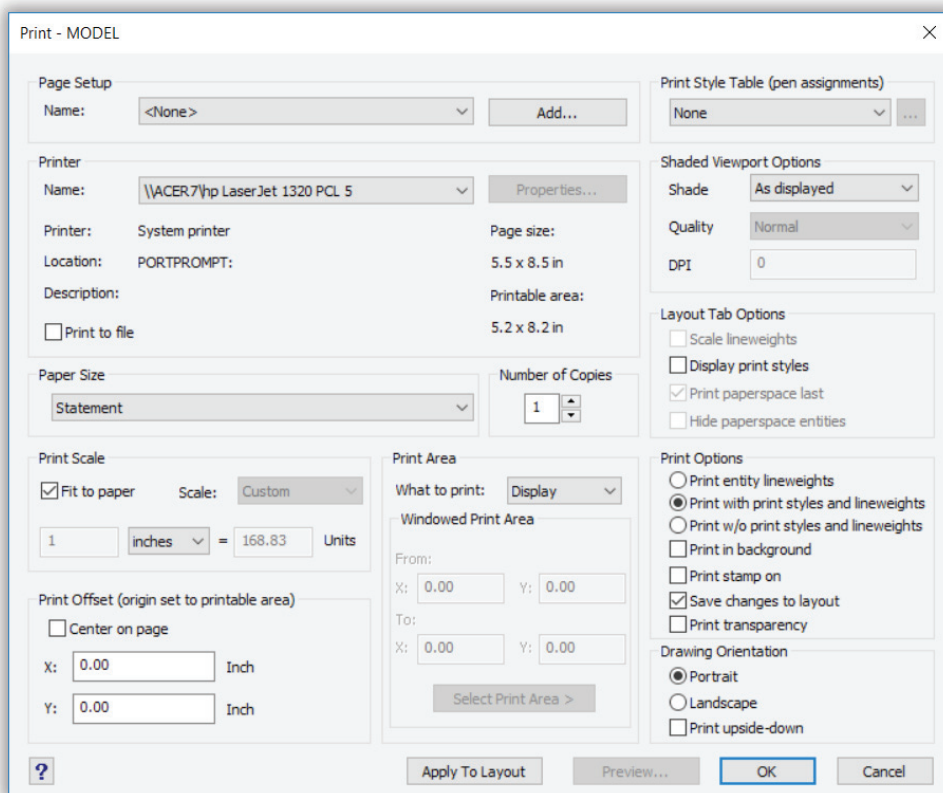
Follow these steps to print a drawing:

1.   Open progeCAD with the drawing you want to print, and then enter the **Print** command:

Command: **print**

(Alternatively, press **Ctrl+P**. as a shortcut keystroke. Note that the Print button on the toolbar executes the QPrint command, discussed later.)
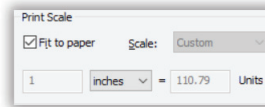
2.   Notice that progeCAD displays the Print dialog box.

In the **Printer** section, choose the printer from the **Name** droplist. Choose any printer that is hooked up to your computer, such as an HP LaserJet or Canon inkjet printer.
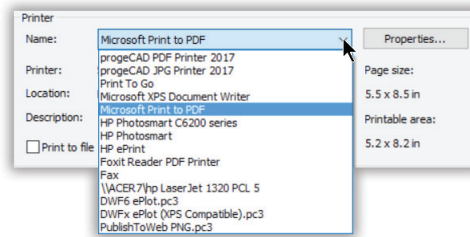
3. In the **Paper Scale** area, turn on the **Fit to Paper** option. This ensures the entire drawing is no larger than the paper in which it will be printed.

4. In the **Print Offset** area, turn on the **Center on Page** option. This centers the drawing on the page, so that it isn't off to one side or another.
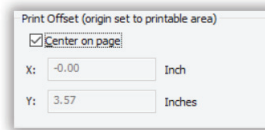
5. In the **Print Area** section, change **What to Print** to "Extents." This setting ensures that everything in the drawing is plotted, and no part of it is cut off.
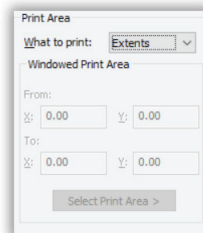
6. That's it for the settings. All the others can be ignored at this point. But, before plotting, always, *always, _always_* take a look at the preview. It takes but a few seconds and ensures the plot will turn out as you expect.

Trust me, plotting has so many options that there are bound to be one or more that are set incorrectly. Preview lets you catch the mistakes before committing time, paper, and ink.

When you click the **Print Preview** button, progeCAD immediately shows the floor plan as it will appear on the printed page.

7. If it looks satisfactory to you, click **Print**. progeCAD sends the drawing to the printer. Wait a few moments for it to appear.

If you don't like the arrangement or size, click **Print Settings** to return to the Print dialog box.

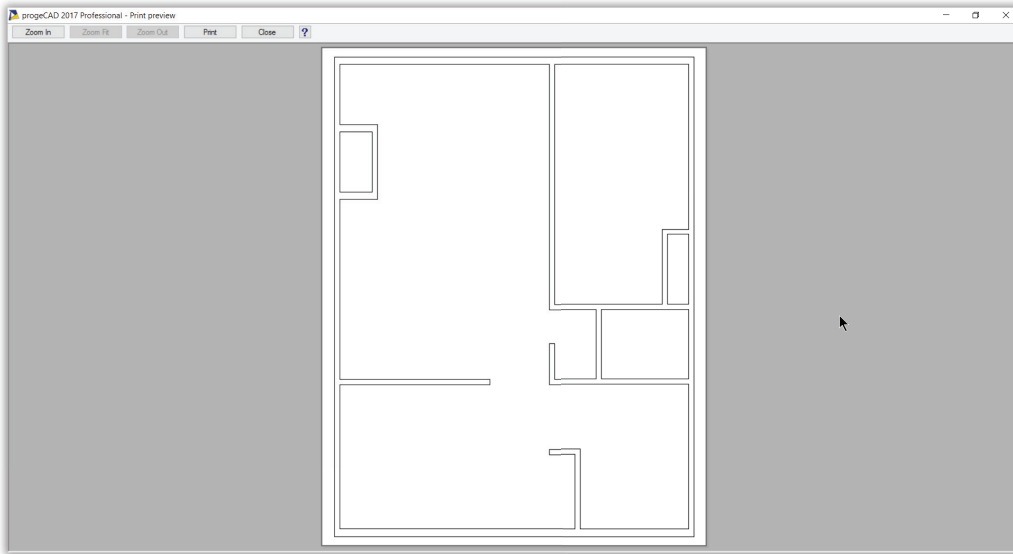---

QPRINT COMMAND

Plots the current drawing using default values and printer.

Command: **qprint**

Toolbar: **Standard | Print**

ALTERNATIVE

Print displays dialog boxes for controlling the plotting process.

Quick Prints (QPrint)

For quick plots, enter the **QPrint** command (short for "quick print"), or else click the **Print** button on the Standard tool bar.

Command: **qprint**

progeCAD immediately sends the drawing to the printer or plotter.

The printing process itself is no any quicker than with the **Print** command; the difference is there is no Print dialog box and no print preview.

When you enter the **QPrint** command, progeCAD quietly reads the settings of the Print dialog box and uses the default Windows system printer. Then it prints the current viewport according to those settings. (The very earliest releases of AutoCAD had a similar QPlot command, but it was removed with v2.0.)

(This is identical to the operation in Word: click the printer icon on the toolbar, and Word sends the document to the printer using the default settings; select the Print item from the File menu, and Word displays the Print dialog box.)

You can get a similar effect with the **Preview** command: after progeCAD displays the preview window, click the **Print** button. progeCAD uses the stored print settings to control the plot.

Preview Prints (Preview)

As I noted above, the **Preview** command lets you see what the plot will look like before committing to printing it. It operates similarly to the print preview feature in word processors and other such software.

1. Enter the **Preview** command.
Command: **preview**

2. progeCAD displays the drawing in the preview window. Along the top of the window are buttons for controlling the preview:

PREVIEW COMMAND

Previews drawings before they are plotted.

Command: **preview**

Aliases: **pre, makepreview, ppreview**

Menu: **File | Print Preview**

Toolbar: **Standard | Print Preview**
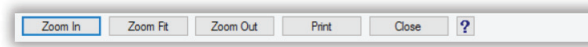
Zoom In enlarges the drawing. Alternatively, you can click the cursor on the drawing. Each time you click, progeCAD zooms in some more. The zoom is centered around the point where you click.

Zoom Out makes the drawing smaller so that you see more of it.

TIP After you click five time to zoom in, progeCAD zooms the drawing out to again fit the preview screen.

Print closes the preview window, and then prints the drawing.

Close closes the preview window, and returns to the drawing editor.

? — displays on-line help.

All About All Printing Options

Let's return to the Print dialog box and examine the purpose of all of its options.

Page Setup section

You can preset plotting settings as "page setups." Once you set all the parameters, you reuse them in future plotting jobs to save you from going through all them, over and over again!
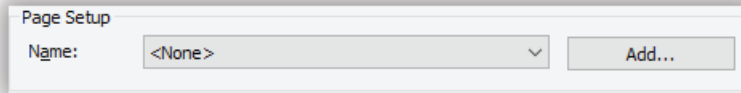
To create a page setup, click the **Add** button, and then give the setup a descriptive name.
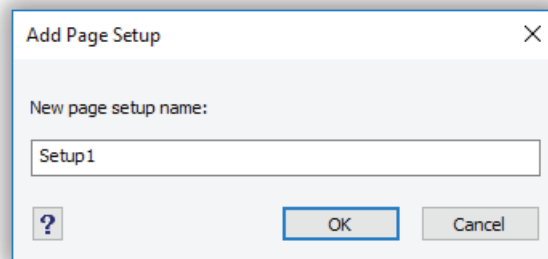
After you click **OK**, progeCAD displays the Add Page Setup dialog box, which looks exactly like the Print dialog box; the sole difference is that you cannot preview or print from it — just set up plotting parameters.

Makes changes to the parameters, and then click **OK**. When you return to the Plot dialog box, click the **Name** droplist to choose from the page setup(s) you saved, and all the settings automatically change.

Printer section

The **Name** droplist specifies the printer to use for the print job.
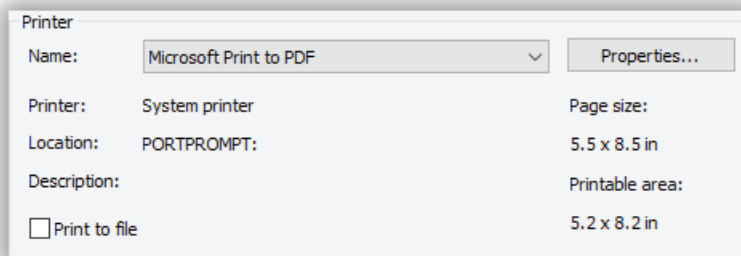
progeCAD lets you use any of the following kinds of printers:

- Any printer set up for your Windows computer
- Printers connected over networks
- Non-printer drivers, such as for fax software and Adobe PDF
- Non-printers specific to Vista, such as XPS Document Writer
- Additional file formats, such as JPG or DWF

The **Properties** button displays the Properties dialog box specific to the printer. This is the same dialog box you would see in any other program.

☐ **Print to file** prints the drawing to a PLT print file. When you choose a file format like DWF as the printer, then this option is turned on by default.

Paper Size section

The **Format** droplist lists all the sizes of media supported by the selected printer. This means that the sizes listed vary, depending on which printer is chosen. The most common size is Letter (aka A-size).
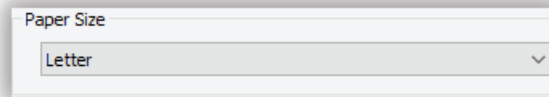
To change the paper sizes listed here, you need to go into the printer's own Properties dialog box, and then see if the list can be edited.

Number of Copies section

The **Number of Copies** text entry box specifies the number of copies to print of the drawing.
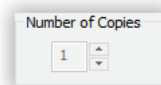
Print Scale section

The **Print Scale** section lets you tell progeCAD how big to plot the drawing.
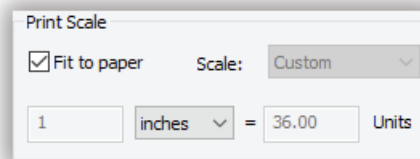
You have two choices: have progeCAD automatically scale the drawing to fit the paper, or else specify the scale factor that progeCAD must use.

☐ **Fit to Paper** automatically adjusts the scale to squeeze the entire drawing to the paper. No matter how big or small the drawing, progeCAD resizes it to fit the paper. progeCAD reports the scale under **Drawing units**, such as 36 units.

When you turn off **Fit to Paper**, then you can set the scale factor:

Scale droplist lets you select a scale factor:
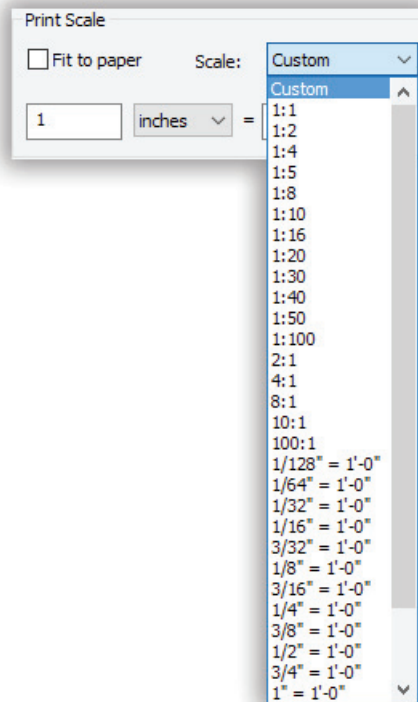
1:1 etc specifies the scale factor for the drawing. Many disciplines work with specific scales, such as 1:50 or 1"=250'. Of the two numbers, the first number is the length on the paper (such as 1"), while the second number is the length in the drawing (such as 250').

TIP   You can change the factor displayed here with the **ScaleListEdit** command, adding scale factors you use and removing those you never need.

From the drop list, you can choose whether drawings are plotted in inches or millimeters, depending on the units of the paper and the drawing:
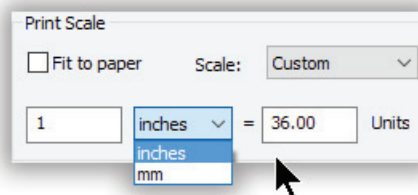
- **Inches** — uses inches (Imperial or British) units for the paper size. This allows you to plot metric drawings on imperial paper, as is common in Canada or USA.
- **Millimeters** — uses millimeters (metric) units for the paper size in Europe.

TIPS The **Paper size** is set by the printer driver, which is accessed through the **Device** tab **Properties** button.

The units of drawings are set with the **Units** command. Use decimal units for metric drawings.

Print Area section

The **What to Print** droplist tells progeCAD how much of the drawing to plot. In most cases, you'll want to select the **Extents** option.
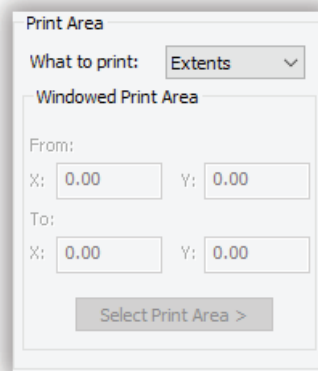
Here is what the options mean:

- **Display** plots only as much of the drawing as is visible in the current viewport. That's exactly the view at the time you entered the Plot command. If the drawing is zoomed in, then only the zoomed-in portion is plotted.
- **Extents** plots the entire drawing. This is the most commonly-used print area option, because it ensures everything is plotted. Everything, that is, except entities on frozen layers. Sometimes, drawings are printed really, really small, and you wonder why. Usually, it's because there are some entities way far, far away from the primary drawing. You can check by performing a **Zoom Extents** before printing.
- **Limits** plots the portion of the drawing within the limits defined by the **Limits** command. The usefulness of this option is not clear to me, because limits are so rarely used.
- **View** plots named views defined earlier by the **View** command. When no views are defined, then this option is not available; if there are, then you can select a view name from the droplist. By defining views ahead of time, you can print specific areas of the drawing, such as a title block or a detail view.
- **Window** plots a rectangular area. You enter either the x, y coordinates of the lower left and upper right corners of the area, or else click the **Select Print Area** button and then pick two points to define the rectangle. This option is useful for printing a portion of the drawing.

Print Offset section

progeCAD normally places the drawing in the lower left corner of the page. The Print Offset area allows you to change this position.
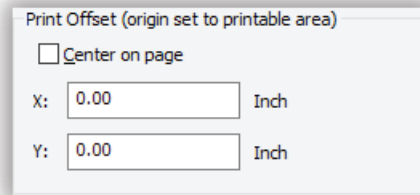
☐ **Center on page** centers the drawing on the page, when on. When off, the drawing is placed in the lower left corner. Frankly, I don't know why this option isn't turned on by default — centered plots look nicer.

X, Y allow you to specify an offset distance. When you plot drawings on preprinted sheets, you sometimes need to offset the drawing from the lower right corner of the page.

Print Style Table (Pen Assignments) section

The Print Style Table section assigns plot properties to entities, such as width, screening, and line weight. Don't worry if you see "None": it means that a plot style has not yet been applied to the drawing. Indeed, no plot style need ever be applied; you just use plot styles when you want extra fine control over the look of the printed result.
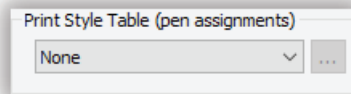
Just as dimension styles affect the look of dimensions, plot styles affect the look of plotted drawings. They allow you to make the look of drawings seen on the screen independent of their look on paper. As I noted earlier, there are two ways to assign plot styles: by colors, or by entities.

Color-based styles map the colors of entities to plotter parameters. In the old days, back when the CAD universe was ruled by $20,000-pen plotters, this was the only way to do it: colors of entities were not meant to identify them in drawings, but to specify how they would be plotted.
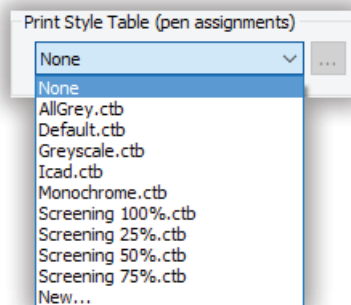
Pen plotters typically had eight pens, usually black and of varying widths, so drafters would work with eight colors in their drawings. When it came time to plot, drafters specified which entity colors would be plotted by which pens. Weird? Yah.

Today's ink jet printers are capable of faking millions of colors and dozens of widths, so pen mapping just isn't needed anymore. But if you need to, here's how: the drafter might decide that all red entities should be plotted with the plotter's pen #1. Pen #1 could be red, but more often it would be black and a specific width. Once colors were mapped between entities and plotter pens, the settings can be saved to *.ctb* (**c**olor **tab**le) files for later reuse.

Today, drafters don't worry about matching pens; red entities are plotted red, and wide entities are plotted wide.

Entity-based styles map entities or layers to plotter parameters. When plot styles are enabled, the Explorer-Layer window displays an extra column so that you can assign plot styles. Settings are saved to *.stb* (**s**tyle **tab**le) files.

See later in this chapter for more about plot style tables.

Print Options section

Finally, a few last options:
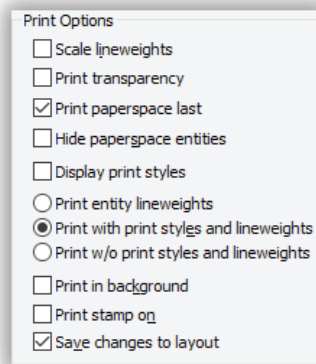
- ☐ **Scale lineweights** — resizes lineweights to match the scale at which the drawing is printed.
- ☐ **Print transparency** — simulates transparent entities in the plot; this options can consume a lot of memory and processing power, slowing down the plot.
- ☑ **Print paperspace last** — prints the model layout first, followed by paper space layouts.
- ☐ **Hide paperspace entities** — does not print entities in paper space.
- ☐ **Display print styles** — displays print styles in paper space.
- ○ **Print Entity Lineweights** — when on, tells progeCAD to send the printer wider lines, as defined by line type widths in the drawing. Generally, this option should be turned on.

- ⊙ **Print with Print Styles and Lineweights** — uses print styles and line weights.
- ○ **Print without Print Styles and Lineweights** — turns off print styles and line weights.
- ☐ **Print in Background** — prints the drawing as a background task.
- ☐ **Print stamp on** — adds plot stamp text to the edge of the drawing; see below.
- ☑ **Save Changes to Layout** — saves the settings to the layout on a layout-by-layout basis. Here "layout" also refers to the Model tab.

Print Stamps

When you turn on the Print Stamp On option, the ⬚ **Browse** button appears. Click it to access the header and footer options.
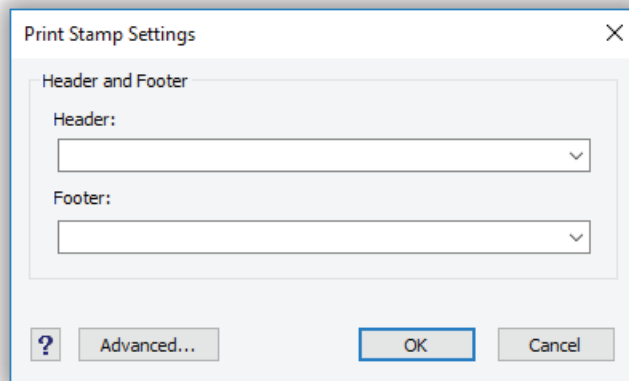
Plot stamps place useful identifying text along the top (*header*) and bottom (*footer*) of plots. The header and footer information can be anything you might find useful, such as the date of the plot and the file name. Simply type text in the text fields, or else choose field text from the droplist:
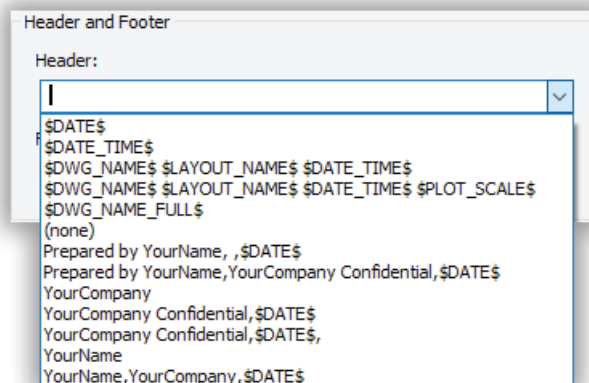
progeCAD includes some preset text, such as today's date, the name and company of the registered user, and the word "Confidential." To see what the header and footer will look like in your plotted drawing, enter the **Preview** command.
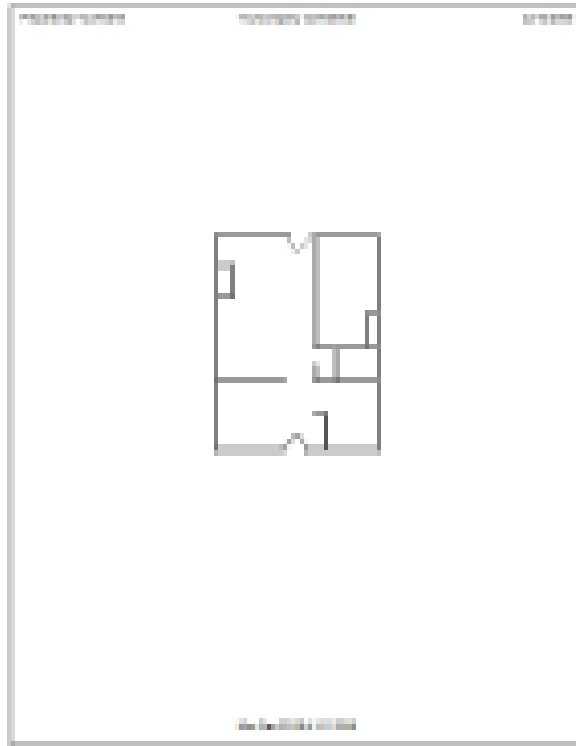
Drawing Orientation section

The **Drawing Orientation** radio buttons switch between Portrait (upright) and Landscape (sideways). This does not change the paper going through the printer; instead, this option send the drawing rotated by 90 degrees to the printer. You change this option when you want the drawing fit the paper better.
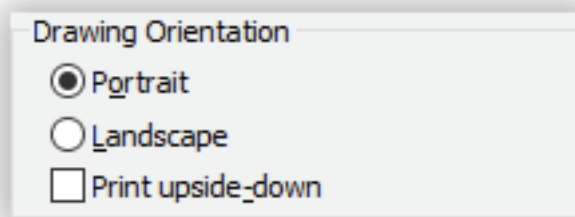
☐ **Print upside-down** — turn on this option when you use preprinted paper and it is inserted the wrong way in the printer.

TIP   To produce mirrored plots, first mirror the entire drawing by pressing **Ctrl+A** followed by **Mirror**. Plot the drawing, and then use the **U** command to un-mirror the drawing.

Other Buttons

The bottom of the dialog box contains a row of useful buttons:
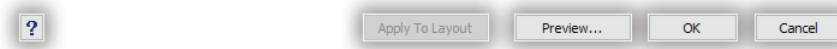
> **?** displays online help.
>
> **Apply to Layout** saves the options to the current layout; the next time you print this layout, the options are restored.
>
> **Preview** switches to the preview window; see Preview command.
>
> **OK** prints the drawing.
>
> **Cancel** closes the dialog box without printing.

About Plot Style Tables

When drafters want fine control over plots, they use *plot style tables*.
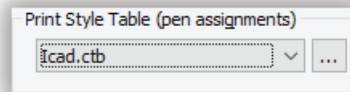
To see a sample plot style, select *icad.ctb* from the **Name** droplist, and then click ... **Browse.** In the Print Style Table Editor dialog box, choose the **Form View** tab.
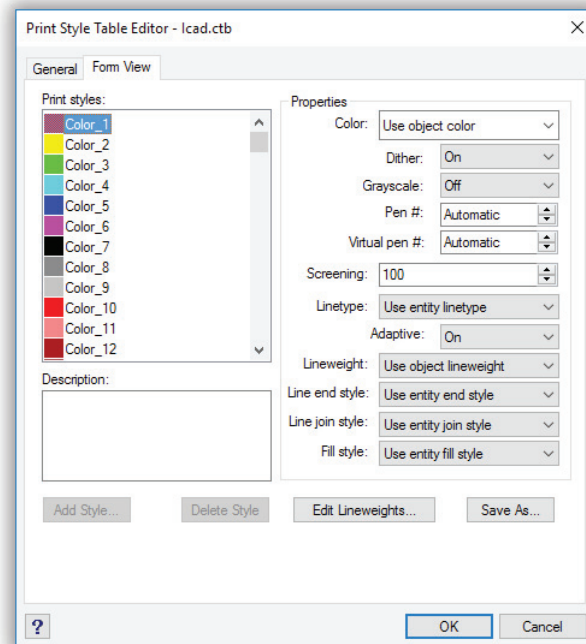
On the left under **Print Styles** are progeCAD's standard 255 colors. On the right are the properties that can be assigned each color.

When a drawing is set up to use plot style tables, the dialog box looks very similar. The differ-
ence is that you see the names of plot styles, instead of the 255 entity colors. Each table can
contain one or more styles, as illustrated below.
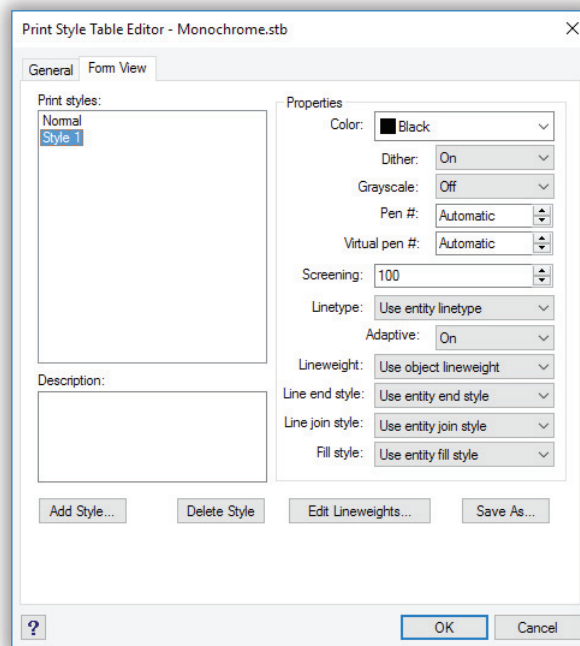
Color overrides the entity color with another color. To plot with the drawing's own
colors, select "Use entity color."

Dither

Grayscale

Pen # specifies which pen should plot the entity; range is 1 to 32. To let the printer
determine the pen number, choose "Automatic."

Virtual Pen # specifies a virtual pen number. Laser printers support the concepts of
virtual pens, because they don't actually have pens. Range is 1 to 255.

Screening specifies the amount of ink used for drawing objects. When set to a value
less than 100%, the plotter uses many fine dots to print objects, instead of printing
them in solid color.

Linetype overrides the entity line type with another line type. For example, you may want to plot the entire drawing using dashed line types. To plot with the drawing's own line types, select "Use entity linetype."

Adaptive determines whether line types are adapted to the length of entities. When this option is turned on, line types are forced to fit evenly between endpoints.

Lineweight overrides the entity line weight with another line weight. For example, you may want to plot the entire drawing using a single line weight. To plot with the drawing's own line weight, select "Use object lineweight."

Line End Style specifies how the ends of lines (and other open objects) are plotted.

Line Join Style specifies how the corners of lines are plotted.

Fill Style specifies the fill pattern for objects. This options can be used in place of the Hatch command.

Add Style adds a new style name to the Print Styles list, whose properties can then be edited with this dialog box.

Delete Style removes the selected style name from the list.

Edit Lineweights adds and removes lineweights from the list of available ones.

Save As saves the settings in *.stb* files.

Electronic Plotting

Drawings can be plotted "electronically" with the **Plot** and **Export** commands. These saves drawing files in formats suitable for display in Web pages and delivery by email, such as PDF and DWF.

Plot to PDF Format

PDF is short for "portable document format," a format created by Adobe for viewing documents. In recent years, it became the most popular way to view and transmit CAD drawings. To view *.pdf* files, you need viewing software, which usually is free, such as Acrobat from www.adobe.com/acrobat.

To export drawings in PDF format, follow these steps:

1. Start the **Plot** command.
2. In the Print dialog box's Print tab, click the **Printer** droplist, and then choose "ProgeSoft PDF Printer."
3. Change the options in the dialog box, as required.
4. Click **Print**. Notice the Select PDF File Name dialog box.
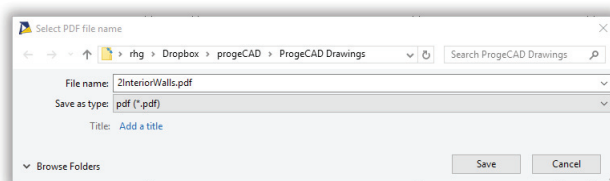
5. Enter a file name. If necessary, change the folder and drive in which to store the PDF file.
6. Click **Save**. progeCAD generates the PDF file, and then asks if you wish to see it.
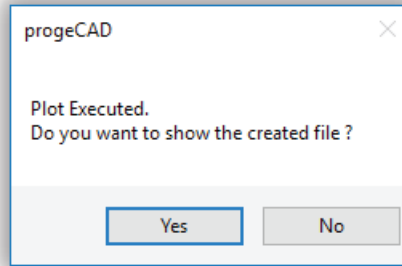
7. Click **Yes**, and the file is opened in the PDF viewing software installed on your computer.

Alternative PDF Exporting

A second way to export drawings in PDF format is with the **ExportPDF** command. The difference from the Plot command is the dialog box in which you specify settings:

Plot command — displays settings more suitable for plotting drawings

ExportPDF command — displays settings more suitable to creating PDF files

Which you use is up to you.
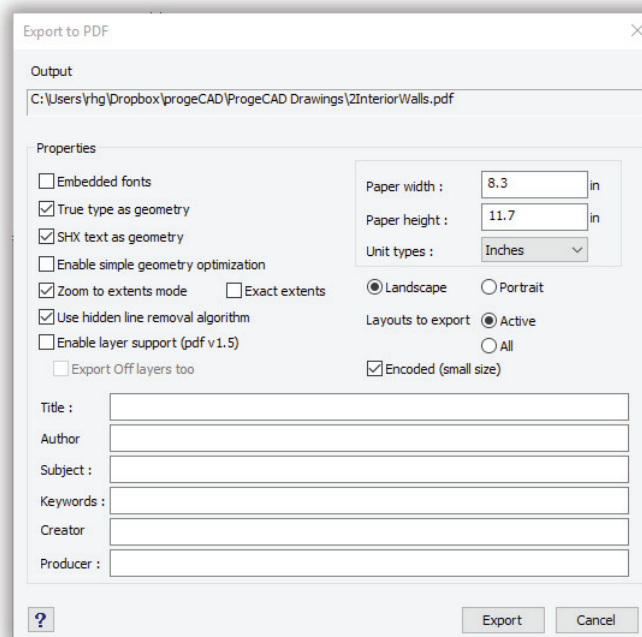
☐ **Embedded Fonts** — includes copies of TrueType font files used by the drawing in the PDF file. This ensures the other people viewing the file will see text correctly, should their computer not have the same fonts as yours.

☑ **TrueType as Geometry** — converts TrueType text to geometry (lines and arcs), instead of saving text as fonts. This may be required when the TrueType font files are covered by copyright and cannot be shared with others.

☑ **SHX Text as Geometry** — converts SHX text to geometry (lines and arcs), instead of saving text as fonts.

☐ **Enable Simple Geometry Optimization** — converts overlapping and connected geometry as single entities. For example, arcs are converted to Bezier curves instead of a series of short line segments. Normally this is a good thing, but older versions of PDF viewing software might not support the geometric elements.

☑ **Zoom to Extents Mode** — zooms the drawing to its extents to maximize the size of the drawing in the viewing window.

   ☐ **Exact Extents** — adds a bounding box to the drawing's extents.

☑ **Use Hidden Line Removal Algorithm** — removes lines that are hidden from view.

☐ **Enable Layer Support (pdf v1.5)** — exports layer names, so that people viewing the PDF can turn layers on and off. As indicated by the parenthetical note, this works in PDF v.5 or later.

   ☐ **Export Off Layers Too** — includes layers that are turned off.

Paper Width — specifies the width of the page in the PDF file.

Paper Height — specifies the height of the page in the PDF file.

Unit Types — specifies whether units are inches or millimeters.

⊙ **Landscape** — places the drawing on the page in landscape (wide) orientation.

   ○ **Portrait** — places the drawing on the page in portrait (tall) orientation.

⊙ **Layouts to Export** — determines if only the current (active) layout is exported, or all are exported.

☑ **Encoded (Small Size)** — compresses the PDF file to make it smaller.

Title, Author, Subject, Keywords, Creator, and Producer — optional descriptions for who and why the PDF was generated.

Export to DWF Format

DWF (design Web format) was developed by Autodesk in the late 1990s as a way to efficiently transmit drawings by email. The format strips out all unnecessary information and greatly compresses what remains to create replicas of drawings that are as much as 100x smaller.

As with PDF files, you can export drawings in DWF format using the Plot command or the DwfOut command, and the difference between the two is the same. Using Plot lets you greater control over the output, while DwfOut provides only a few options specific to the format.

To create DWF files of drawings, follow these steps:

1. Enter the **DwfOut** command.

   Command: **dwfout**

2. In the Export Drawing As dialog box, give the file a name, or accept the default, which is the drawing's name with the *.dwf* extension. Click **Save**.

3. Notice the DWF File Version and Format dialog box.
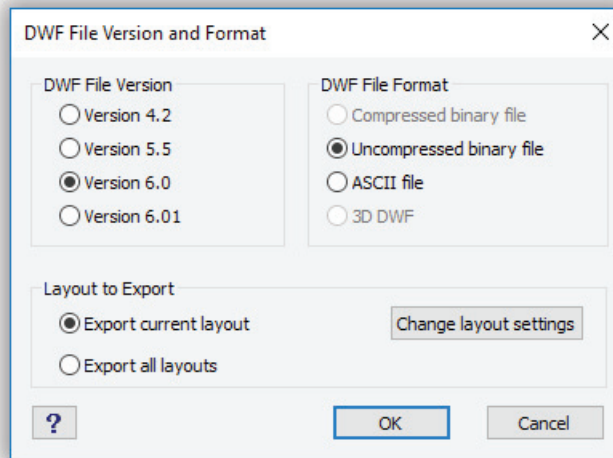
Choose from the following options:

- **DWF File Version** — you should use the newest version (such as v6.01), except when recipients cannot read the file because they are using old versions of the viewing software.

- **DWF File Format** — Compressed Binary File produces the smallest file format.

- **Layout to Export** — choose Export Current Layout or Export All Layouts, depending on how much you want to include with the file. Note that "layout" refers to both model tab and to layout tabs.

  Change Layout Settings — brings you to the Modify Page Setup dialog box, as described earlier in this chapter. It lets you specify options as in the Plot dialog box.

4. Click **OK**, and progeCAD exports the drawing; look for it in the folder specified by the Look In droplist.

The files can be sent by email, added to a Web page, or viewed with viewing software. To view *.dwf* files, download the free DWF Viewer software from www.autodesk.com/dwf.

Note that Autodesk appears to be deprecating DWF in favor of PDF.

progeCAD also exports drawings in other useful formats, although some are limited to being used with 3D models; they cannot be used with 2D drawings.

Format	2D/3D	Exports
DWFx	2D and 3D	Compressed version of DWF
DXF	2D and 3D	Drawing interchange format
DGN	2D and 3D	Microstation design files
STL	3D only	3D models to 3D printers
SAT	Limited	3D solids and 2D regions to ACIS format
PDF	3D only	3D PDF format embeds models in PDF files
3DS	3D only	3D models to 3D Studio rendering software
POV	3D only	3D models to POV Ray rendering software
LWO	3D only	3D models to LightWave rendering software
DAE	3D only	3D models to Collada rendering software
WMF	2D and 3D	Drawings in Windows Metafile Format
EMF	2D and 3D	Drawings in Enhanced Metafile Format
BMP	2D and 3D	Drawings in raster bitmap format
SVG	2D and 3D	Drawings in scalable vector graphics format

SVG is the approved standard for displaying vector images (i.e., CAD drawings) on the Internet. Although it is an official standard, it isn't used much, although progeCAD can export it through the Export command. Once created, you can view the files with your Web browser. Learn more about SVG at www.w3.org/Graphics/SVG.

Importing Other Formats into progeCAD

progeCAD can read 2D and 3D PDF files other than DWG. Note that the accuracy and content of the imported files might not be as good as DWG files. The **Import** command supports the following formats:

Format	2D/3D	Imports
DWF	2D and 3D	Compressed version of DWF
DXF	2D and 3D	Drawing interchange format
DXB	2D only	Ancient format used with CAD/camera software
DGN	2D and 3D	Microstation design files
DAE	3D only	3D models from Collada rendering software

A second command, **Convert**, gives you some control over how the translation takes place.
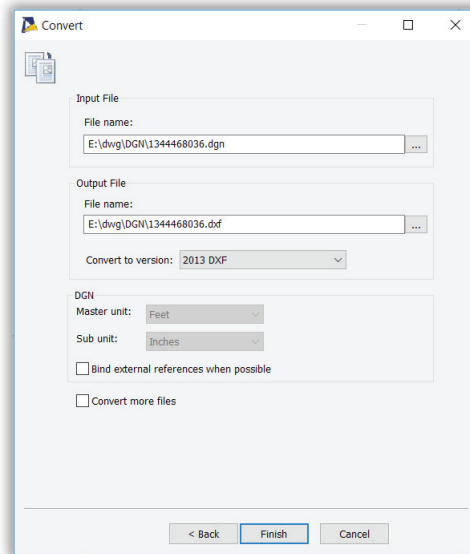
Converting PDF to DWG

progeCAD provides an external utility program, Pdf2Cad, to convert the vector portions of PDF files to DWG drawing files. You can launch the program from inside progeCAD by entering **Pdf2Cad** at the 'Command:' prompt.
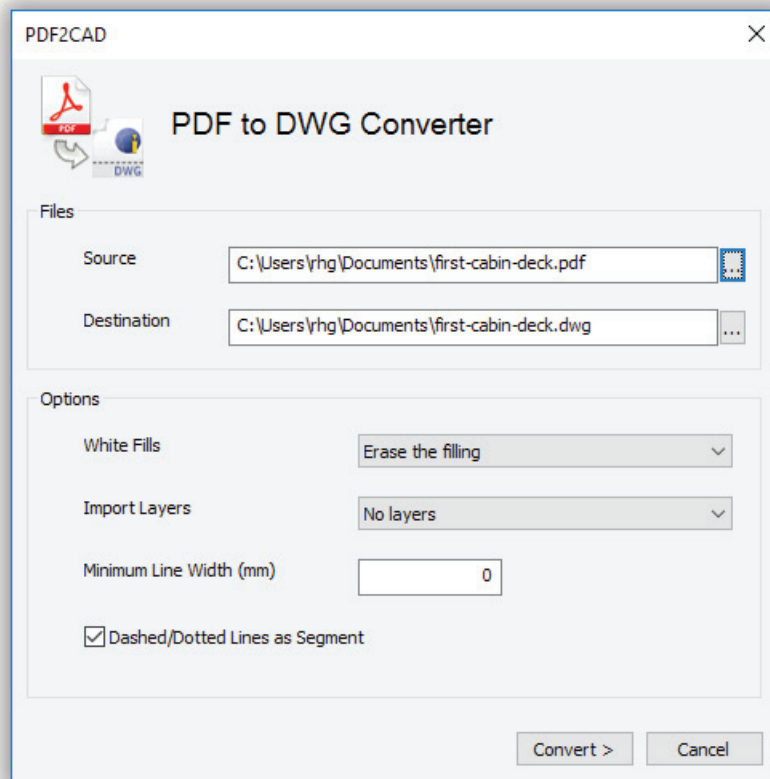
White Fills — determines what happens to filled areas colored white:

- **Maintain the filling** keeps filled areas as they are.
- **Erase the filling** removes fills that are colored white
- **Highlight the filling** places a border around filled areas

Import Layers — doesn't import layers, but rather places imported entities onto layers that depend on their properties:

- **No Layers** places all entities on layer 0
- **By Color** segregates entities by their color, such as all entities of color green on a layer that also is colored green.
- **By Line Width** segregates entities by their color
- **By Layer** imports layer names and properties, if they exist in the PDF file

Minimum line width — specifies the line weight of imported entities.

Dashed/Dotted Lines as Segment — turns dashed lines into individual segments

After conversion is complete (large PDF files can take a while), progeCAD opens the drawing for you to view.

Vector entities are converted to CAD ones as much as possible. For instance, text becomes mtext, lines become polylines, but hatches become individual lines and dimensions become a combination of mtext, polylines, and filled areas (arrowheads). Raster images are converted to images. Only the first page of a multi-page PDF document is converted.

Sending Drawing Packages (eTransmit)

With the help of the **eTransmit** command, you can send a drawing and all of its related support files in a single package for transmission over the Internet or on a USB thumbdrive.

When sending drawing files to other users , it is important to attach dependent files, such as xrefs, images, and fonts. If these are missing, then it becomes impossible for the recipient to view and edit the drawings correctly.

The eTransmit automatically includes dependent files in a transmission package, reducing errors down the road. If you wish, you can manually add other files to the package, such as documents, spreadsheets, and other drawings. You can also remove files from the package that eTransmit automatically added, such as those that might be restricted due to reasons of copyright or confidentiality.

To get a drawing ready for transmittal, follow these steps:

1. Enter the **eTransmit** command.

2. Notice that the Create Transmittal dialog box lists the name of the current drawing file (*5Dimensionsl.dwg*, in the figure below) and names of dependent files, *pcad.fmp (the font mapping file)* and *simplex.shx* (a font file).
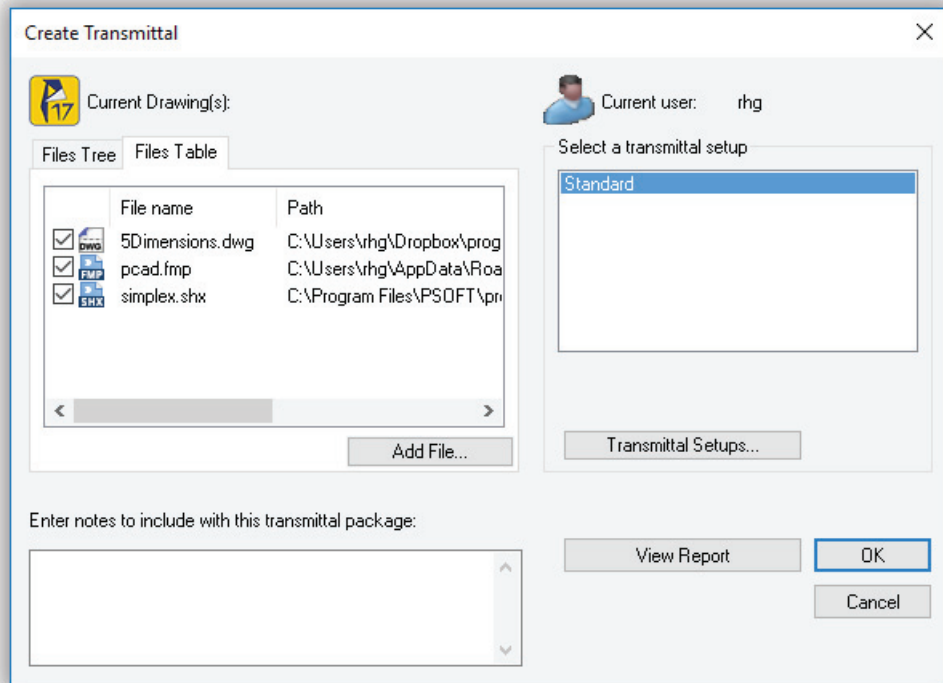
The Create Transmission dialog box contains two tabs that let you view and change the files to be added to the transmission package:

Files Tree lists files in order of dependency. You can expand each drawing file to view the related dependent files that progeCAD adds automatically to the transmission package. To remove dependent files that you don't want to include in the transmission package, click the check boxes to remove the check mark.

Files Table lists files in alphabetical order. This is a summary table that presents the names files, the paths of relative folders, and file details. You can select and deselect each file by clicking the check boxes.

3. To add files, click the **Add File** button. progeCAD displays the Add File to Transmittal dialog box. Choose a file, and then click **Open**. Repeat to add additional files.

4. **Enter notes to include with this transmittal package** are for the persons who receive the transmission package. (Later, eTransmit will generate a report file automatically that contains the list of the files present in the transmission package.)

5. The contents of this report are displayed when you click the **View Report** button.
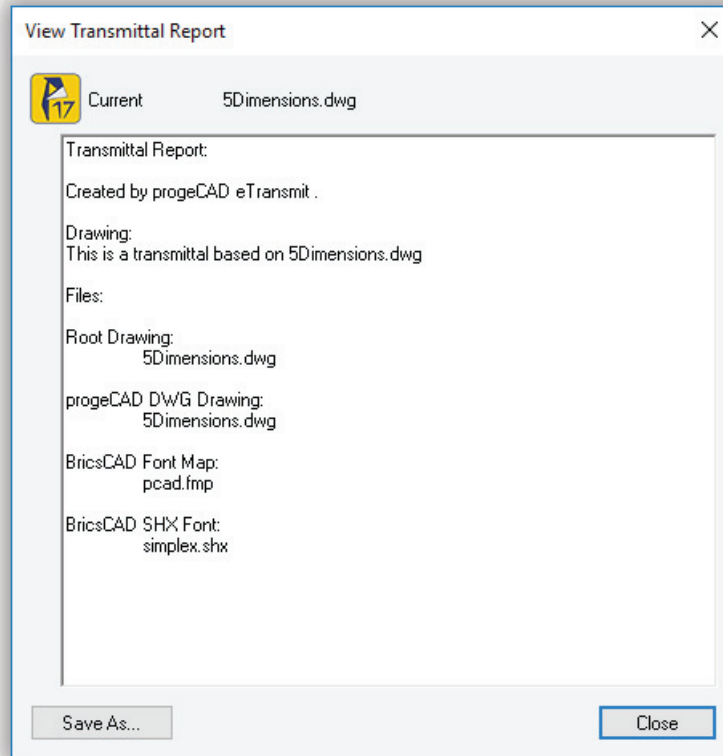
6. You further fine-tune the transmittal with the **Transmittal Setups** options.

Transmittal Setups

Transmittal setups allow you to change settings of transmission packages.

If you need to send transmission packages several times within one project, it is better to save it as a transmittal setups. The default setup is Standard. The Create Transmittal dialog box shows you a list of saved setups from which you can choose each time when you transmit a group of files.

To create new transmittal setups, follow these steps:

1. Click **Transmittal Setups**.

2. Notice the Transmittal Setups dialog box. It lists the names of saved setups, and lets you rename, modify, or delete them. (You cannot rename or delete the default setup, named "Standard.")
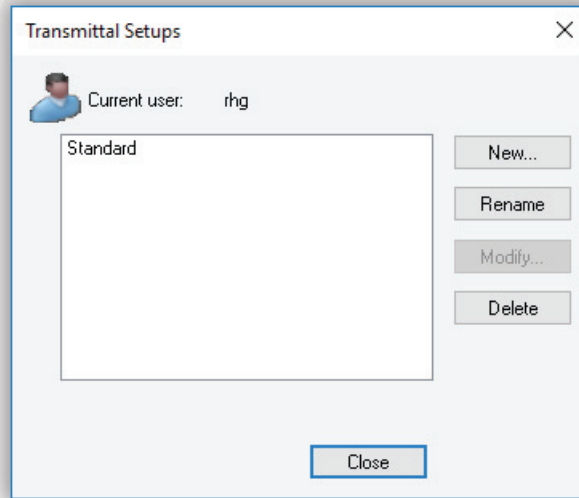
3. Click **New** to create a new setup.

4. Notice the New Transmittal Setup dialog box. Enter a name that describes the setup.
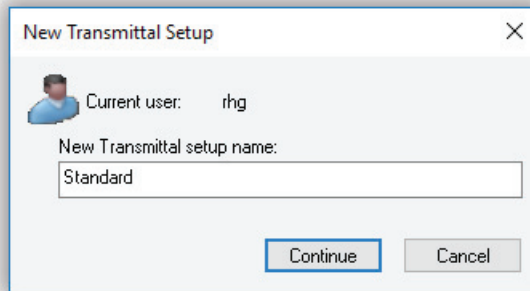
Click **Continue**.

5. Notice the Modify Transmittal Setup dialog box. The settings in the dialog box allow to you to specify the following options:
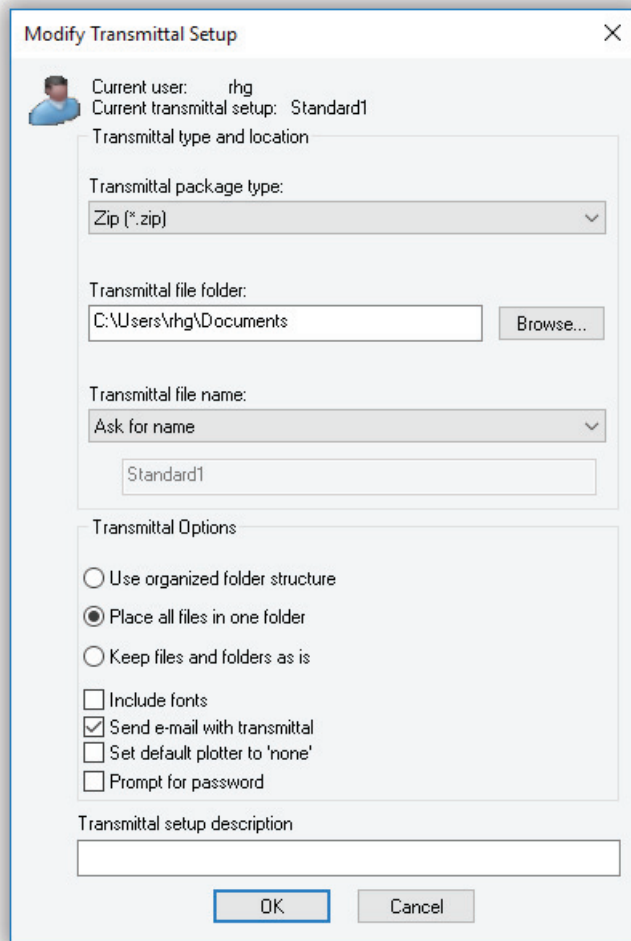
Transmittal package type — compresses the package in either a ZIP file or in a folder.

Transmittal file folder — names the folder in which to place the files.

Transmittal file name — provides the name for the transmittal package, other right now, or else later when the files are saved.

Use organized folder structure — the structure of the files should be organized according to a logical hierarchy, or else inserted all in a single folder, or else copied to the addressee's computer as-is. If a FTP or HTTP destination is specified, the transmission package uses the single folder option.

Include fonts — places the TTF and SHX font files used by the drawing in the transmittal package.

Send email with package — opens the email client automatically, and attaches the transmission package.

Set default plotter to 'none' — removes references to your plotters so that the recipient's CAD program doesn't get confused over plotter assignments.

Prompt for password — adds password protection to the transmission package. The password must be sent to the recipient separately.

Transmittal setup description — describes the particulars of this setup.

6. Change options as necessary, and then click **OK**.

7. Notice that the Transmittal Setups dialog box now contains the new setup. Click **Close**.

8. In the Create Transmittal dialog box, choose the name of a transmittal setup, and then click **OK**.

   If you chose the "Ask for name" option in the transmittal setup, then progeCAD prompts you for the file name and folder at this point.
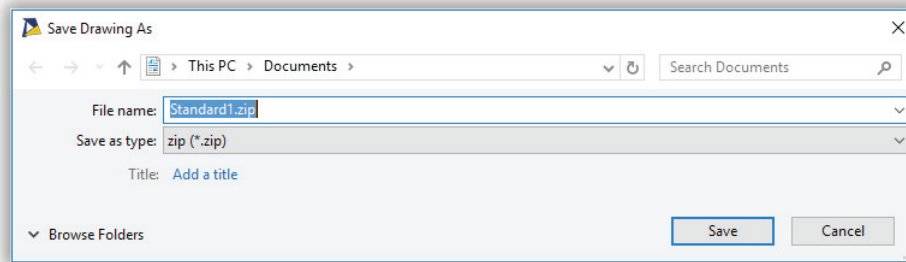
If you chose the "Send email with transmittal" option, then progeCAD creates a new message with your default email software, and adds the transmittal package as an attachment.

After creating the transmission package, you can make it available through the Internet, such as via Dropbox, or send it to other users as an email attachment. If you send the transmission package via email, there is an option in the Modify Transmittal Setups dialog box that launches the default system email application automatically, and attaches the transmission report file.

TIP Full (absolute) paths of dependent files are converted to relative paths or "no path" to ensure that the drawing file can identify the dependent files.

Creating Symbols

With the walls in place, it's time to populate the interior of the apartment. You need to add doors and windows, as well as furnishings. In this chapter, you create a door and a window from scratch, and then turn them into blocks (symbols), which are placed into walls. Later in this chapter, you learn how to insert ready-made symbols of furnishings and how to access symbol libraries on the Internet.

The fixed amenities include the bathroom fixtures, kitchen arrangement, and the fireplace. These remain part of the unit when it is purchased and sold.

Non-fixed amenities are provided by the purchaser; they are shown only to give an idea of how parts fit in the living spaces. These include the living room furniture, dining table and chairs, and bed.

In This Chapter

- **Rectangle** command
- **Insert** command
- **WBlock** command
- **Arc** command
- **-Insert** command
- **iCADLib** command
- **Block** command
- **ExpBlocks** command

Create Door and Window Symbols

When you need to draw the same kind of thing over and over, it makes sense to use blocks. *Blocks* are symbols. Floor plans use, for instance, lots of doors. There is no point drawing a door over and over again, when it is possible it place an entire door at one go. Looking at our floor plan, it looks like about eight doors are needed.

Instead of drawing the eight doors,you draw one, and then make copies. You could use the Copy command, but doors are made of several lines and arcs, and it can be tricky copying groups of lines. It's better to turn the group into a single entity. You do this with the **Block** command.

To plan the doors that are needed, I printed out the drawing, and then sketched doors where they are needed.
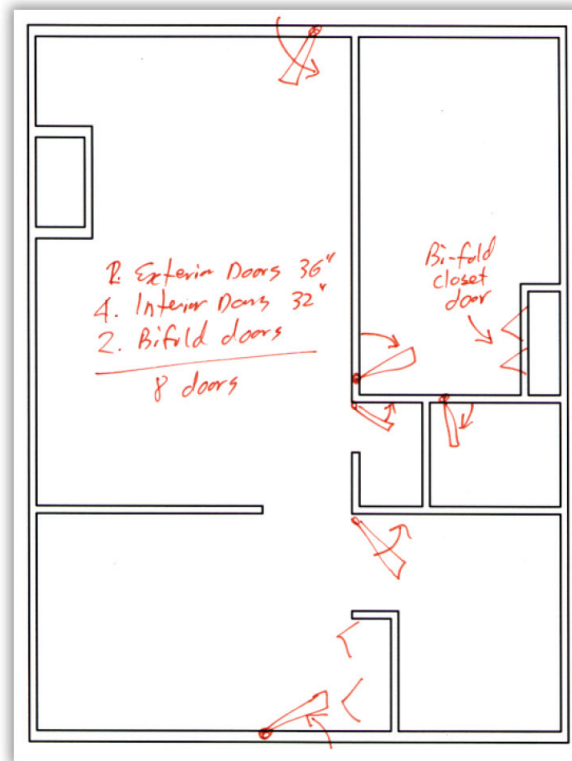
Looking closely at the sketch, there are at least two kinds of doors: regular doors for entering rooms, and bi-fold doors for closets. Among the set of entrance doors, there are two types: exterior doors that are larger (36") to make it easier to move furniture in and out, while interior doors are usually smaller (32") to consume up less wall space.

In summary, the following types of door are needed:

Type of Door	Number Needed
36"-wide exterior door	2
32"-wide interior door	4
Bi-fold door	2

That means you need three blocks, one for each type of door. Creating and placing symbols takes these steps:

1. Draw the symbol with the **Line**, **Arc**, and other drawing commands. (In this book, I use the term *symbol* for the group of entities before they are turned into blocks.)
2. Convert the symbol into a block with the **Block** command.
3. Insert the block into the drawing with the **Insert** command.

(If you had problems completing the previous tutorial, the exterior walls drawing is available as *2InteriorWalls.dwg*.)

RECTANGLE COMMAND

Draws rectangles.
Command: **rectangle**
Aliases: **rectang, rect, rec**
Menu: **Insert| Rectangle**
Toolbar: **Draw 2D | Rectangle**

ALTERNATIVES

Polygon draws regular polygons.
Line draws lines.

Step 1: Draw Door Symbols (Rectangle, Arc)

Door symbols consist of thin rectangles that represents doors, and arcs that show the *swing* of doors. Doors are usually shown open at 45 degrees. Symbols are often simplified representations of physical objects, so the door lacks details, such as hinges and knobs.
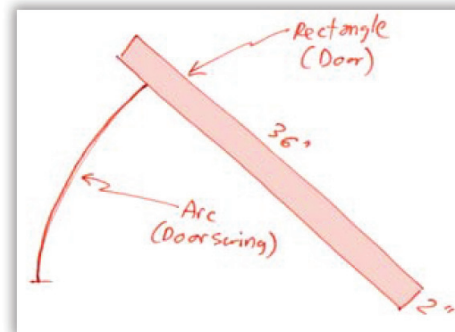
Here's is how to draw the first door, the 36"-wide, 2"-thick exterior door symbol.

1. In progeCAD, press **Ctrl+N** to start a new drawing. (If you are prompted for a template drawing, press **Enter,** or respond with none.)
2. Turn on snap, grid, and lineweight modes by clicking the related icons (or words) on the status bar.
3. To draw the 36" x 2" door rotated at 45 degrees, use the **Rectangle** command:

2. Other corner at 36,2

3. Rotation angle of 45 degrees

1. Start of rectangle at the origin 0,0
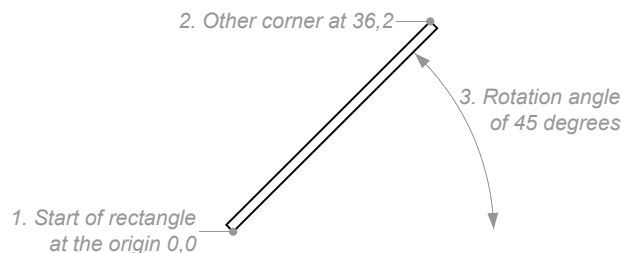
a. Enter **Rectangle** at the command prompt.
   Command: **rectangle**

b. Specify the rotation option, "r". progeCAD will ask later for the angle, after you specify the rectangle's size.

Specify first corner of rectangle or [Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width]: **r**

c. Indicate the two opposite corners of the rectangle:

Specify first corner of rectangle or [Chamfer/Elevation/Fillet/Square/Thickness/Width]: **0,0**

Other corner of rectangle: **36,2**

d. And then specify the angle of 45 degrees:

Rotation angle for rectangle <0.0000>: **45**

e. If you cannot see the entire rectangle, use the **Zoom Extents** command:

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or [In/oUt/All/Center/Extents/Left/Previous/Right/Window/Object/Dynamic]: **e**

<table>
<tr><td>

ARC COMMAND
_____

Draws arcs.

Command: **arc**

Alias: **a**

Menu: **insert| Arc**

Toolbar: **Draw 2D | Arc**

_____
ALTERNATIVES

Polyline draws polylines and polyarcs.

Ellipse draws elliptical arcs.

</td></tr>
</table>

4. Indicate the door swing with the **Arc** command:

a. This arc is defined by its starting point, center point, and angle of rotation:

Command: **arc**

b. Use entity snap modes to start the arc at the end of the rectangle, as follows:

Specify start of arc, ENTER to use last point or [Center/Follow]: **end**

Snap to endpoint of: *(Pick point 1.)*

c. The center of the arc is the origin of the door, (0,0). Use the **Center** option, and then specify the origin, as follows:

Specify second point or [Angle/Center/Direction/End/Radius]: **c**

Center of arc: **0,0**

d. The end of the arc is defined by the angle. progeCAD measures angles counterclockwise, so you need to specify a negative angle of 45 degrees:

Specify end point or [Angle/Length of chord]: **a**

Included angle: **-45**

1. Start the arc with ENDpoint esnap

2. Center of arc at origin 0,0

3. Angle of arc is -45 degrees
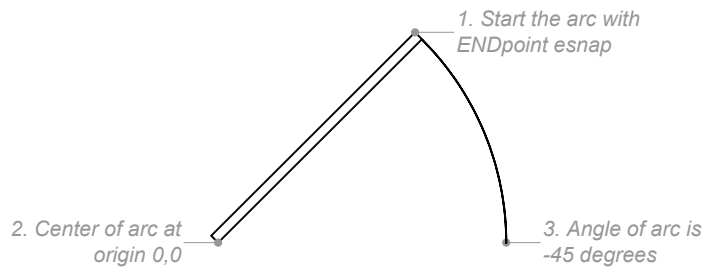
With the symbol drawn, it should look like this:

36"-wide door

Door swing arc

Location of the hinge
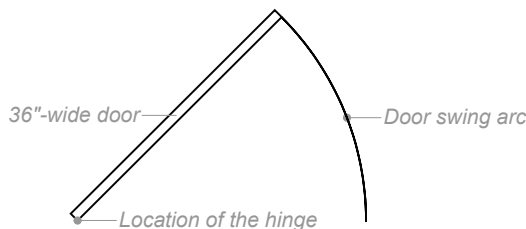
Eighteen Ways to Draw an Arc

Arcs are one of the hardest entities to draw with CAD, because it can be difficult to determine ahead of time how they will turn out. And, because there are 18 ways to draw an arc! In general, progeCAD works with any 3 of the eight parameters illustrated by the figure below:
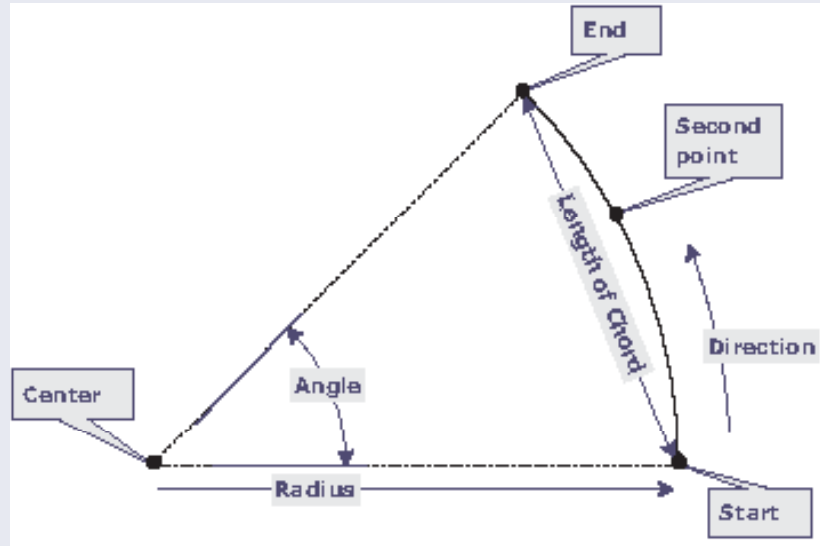

Arc Drawing Methods

The **Arc** command initially prompts you for the location of the arc's center or start points. After that, it prompts you for two other parameters, as listed by the table below. After you draw one arc, progeCAD has two more prompts that connect the new arc to the previous one: **Use last point** and **Follow**.

Initial Prompt	Second Prompt	Third Prompt
Center	Start	Angle
		Length of chord
		End point
Start of arc	Angle	Center
		End point
	Center	Angle
		Length of chord
		End point
	Direction	End point
	End	Angle
		Direction
		Radius
		Center point
	Radius	Angle
		End point
	Second point	End point
Use last point	End of arc	
Follow	End of arc	

The next step is to turn the symbol into a block, so that it can be reused in this and other drawings.

Step 2: Convert to Blocks (Block)

The **Block** command converts a collections of entities into a single entity, called a "block." To create a block, progeCAD needs to know at least three things: (1) the name of the block; (2) its base point; and (3) the objects that make up the block.

Here's how to do it.

1.   Enter the **Block** command:
Command: **block**

Notice the Block Definition dialog box.

2.   Blocks are known by their names. To help sort large collections of blocks, it makes sense to name blocks logically.

For instance, I've decided to name this block "door-36-exterior". Notice that this name goes from the general ("door") to the specific (width and type). In this way, all the door blocks are listed together.

Name: **door-36-exterior**
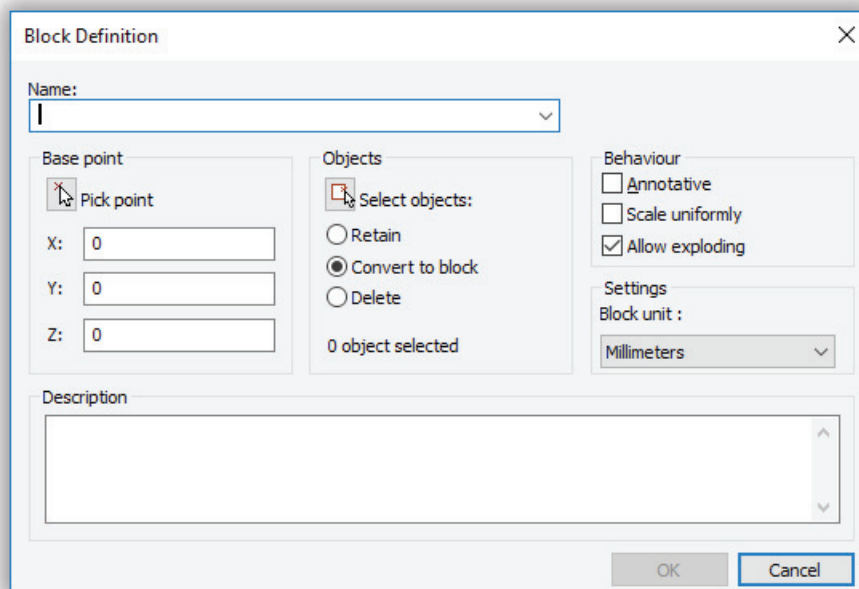
3.   When you later place the block in the drawing, the point you pick is called the "insertion point." You need to tell progeCAD where that insertion point needs to be.

In most cases, like this one, you click the [icon] **Pick Point** button, and then pick a point that makes sense, like a corner or the center of the symbol.

For this door, pick the location of the hinge, which just happens to be at the origin:

X: **0**

Y: **0**

Z: **0**

4. Now you get to select the entities, i.e., the rectangle and the arc.

   a. Click the Select Entities button.

   b. Notice that the dialog box disappears (temporarily), and that progeCAD is prompting you in the command bar:

   Select entities: **all**

   Select entities: *(Press **Enter** to return to the dialog box)*

   Back in the dialog box, progeCAD reports the number of entities it found:

   2 objects selected

5. The last step is to decide on the fate of the symbol:

   • **Retain** — creates the block definition, and retains the symbol in the drawing.

   • **Convert to block** — creates the block definition, and inserts it into the drawing.

   • **Delete** — creates the block definition, and erases the symbol.

   Choose the Convert to Block radio button. The content of the dialog box should now look like this:
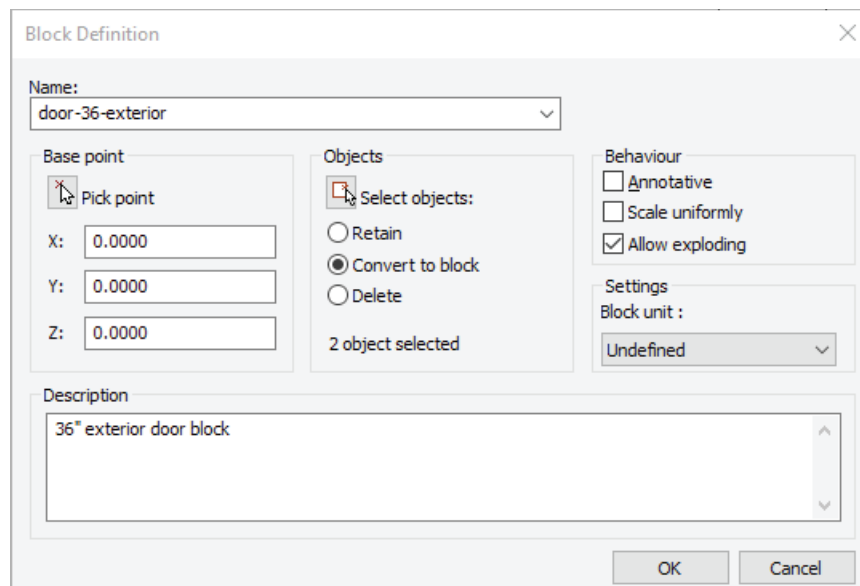
6. Click **OK** to end the command.

7. Save the drawing with the **SaveAs** command, using the name "Door-Exterior-36. dwg."

Sharing Blocks (ExplBlocks)

You created the block in one drawing, but need to use it in another. Sharing blocks between drawings is a easy as copy'n paste. (If you had difficulty creating the 36" exterior door block, you can access it from the *door-exterior-36.dwg* file.)

1. Enter the **ExpBlocks** command to open the Explorer to the blocks section.

2. In Explorer, select the "door-36-exterior" block, and then press **Ctrl+C**. This copies the block to the Clipboard.
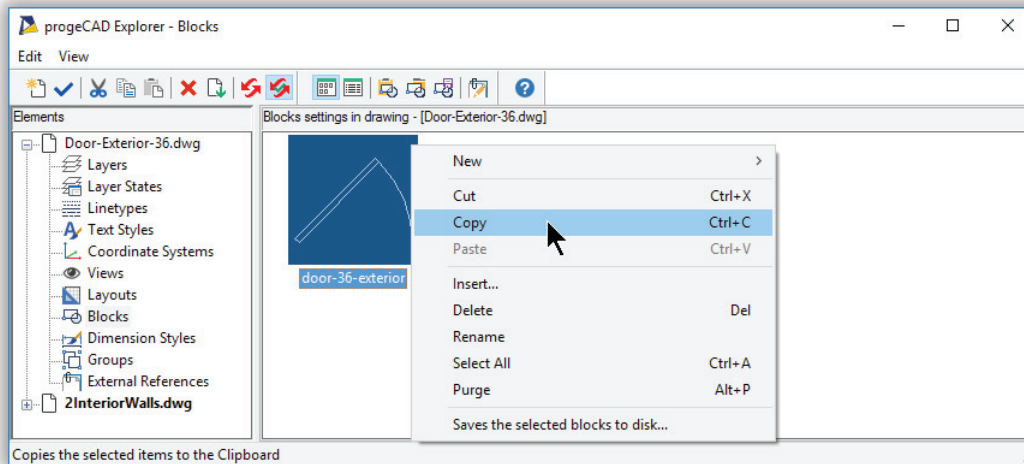
3. In Explorer's Elements pane, select to the *2InteriorWalls.dwg* file.

4. Select **Blocks**, and then press **Ctrl+V**. (Or, right-click and then choose **Paste**.)
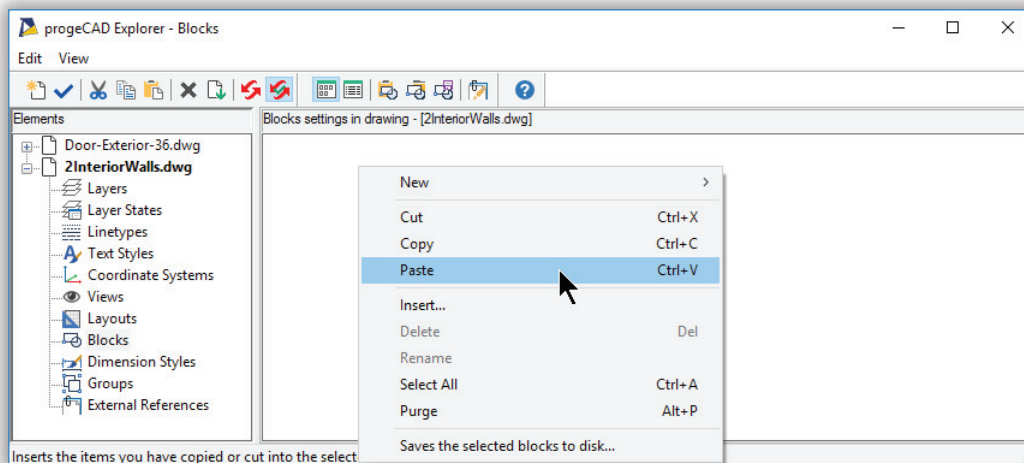
The block definition is added to the drawing, and appears in Explorer.

Step 3: Insert Blocks (Insert)

You now place the door block in the floor plan drawing. Technically, you insert the block *definition* to create block *instances*. There are two ways to place blocks in drawings:

- With the Explorer
- With the Insert command

We'll use both methods, one for each block insertion.

1.   First, in the *2Interior-walls.dwg* drawing, change the working layer to "Assets-Fixed."

2.   Next, plan the location of the first door. It is to be inserted is the entrance from the hallway, the door at the bottom of the floor plan. It's helpful to zoom closer to the area, so use the **Zoom Window** command:

```
Command:  z
Specify corner of window, enter a scale factor (nX or nXP), or [In/oUt/
All/Center/Extents/Left/Previous/Right/Window/Object/Dynamic]:  w
First corner: (Pick a point)
Opposite corner: (Pick a point)
```

3.   On the Explorer's toolbar, click the 🗔 **Insert** button.
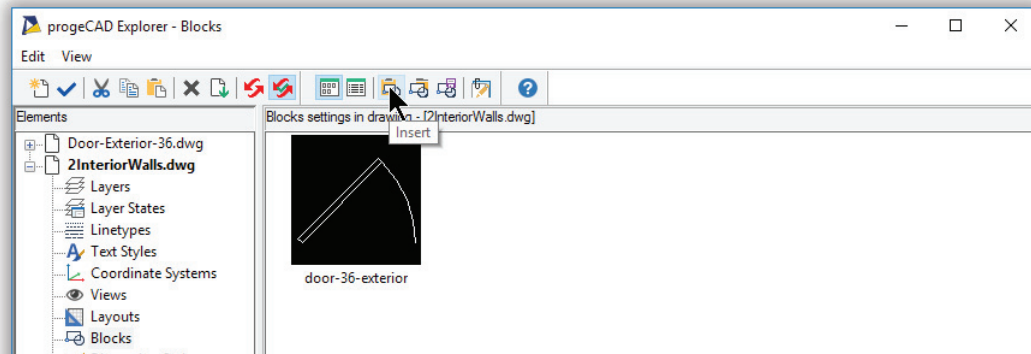
4.   On the command bar, progeCAD prompts you:

```
Command: _insert
Specify block to insert or [? to list blocks in drawing/BROWSE/EXPLORE]: <DOOR-36-EXTERI-
OR>: DOOR-36-EXTERIOR
```

Notice that progeCAD runs the Insert command and fills in the name of the block for you.

5.   Use the **NEArest** entity snap to help you insert the block accurately:

```
Multiple blocks/<Insertion point for block>:  nea
Snap to nearest of: (Pick a point on the wall.)
```

6. You have the option of scaling and rotating blocks during insertion, but this time you don't do this. Press **Enter** at each of the following prompts:

Corner/XYZ/X scale factor <1.000000>: *(Press **Enter** to accept the default value)*

Y scale factor: < Equal to X scale (1.000000)>: *(Press **Enter** to accept the default value)*

Rotation angle for block <0>: *(Press **Enter** to accept the default value)*

7. The door block is inserted. But there is the matter of the door frame: it doesn't exist. Draw the door frame as a pair of lines, using the appropriate entity snaps:

Command: **line**

Specify start point, ENTER to use last point or [Follow]: **int**

Snap to intersection of: *(Pick at the door's hinge point)*

Next point or [Angle/Length]: **per**

Snap to perpendicular of: *(Pick a point on the opposite wall line)*

Next point or [Angle/Length/Follow/Undo]: *(Press **Enter** to end the command)*

INTersection ensap
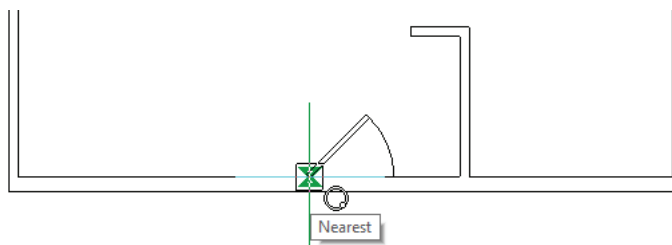
PERpendicular esnap
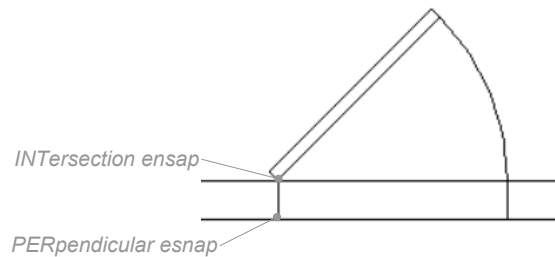
8. Draw the line for other side of the frame:

Command: **line**

Specify start point, ENTER to use last point or [Follow]: **int**

Snap to intersection of: *(Pick at the door's swing arc)*

Next point or [Angle/Length]: **per**

Snap to perpendicular of: *(Pick a point on the opposite wall line)*

Next point or [Angle/Length/Follow/Undo]: *(Press **Enter** to end the command)*

9. Use the **Trim** command to cut away the unneeded portion of the wall.

Command: **trim**

Select cutting entities for trim or ENTER to select all: *(Pick one door line)*

Entities in set: 1

Select cutting entities for trim or ENTER to select all: *(Pick other door line)*

Entities in set: 2

Select cutting entities for trim or ENTER to select all: *(Press **Enter** to end selection)*

Cutting entity —— ——Cutting entity

Entities to be trimmed away
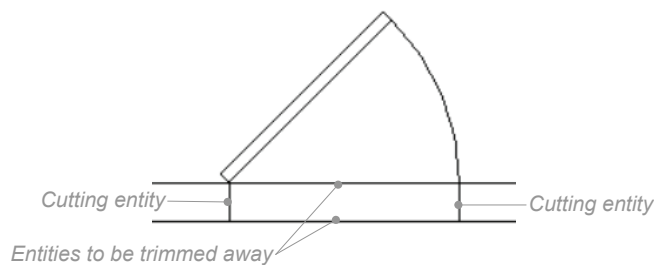
Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Pick one wall line)*

Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Pick other wall line)*

Select entity to trim, shift-select to extend or [Edge mode/Fence/Projection/Undo]: *(Press **Enter** to end command)*
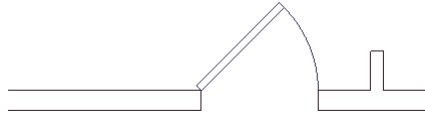
Don't forget to create and insert blocks of the interior doors (32" x 2") and the bi-fold closet doors (24" x 1"). They are both available as *Door-Interior-32.dwg* and *Door-Bifold-48.dwg*.

Inserting Blocks Upsidedown

Now insert the same exterior door block, but at the other end of the apartment: the door to the patio. This door swings in, so that when open, the door is against the living room wall. This time, use the **Insert** command:

1. Start the **Insert** command.

    Command: **insert**

    Notice the Insert Block dialog box. It lets you insert blocks from the current drawing (Block Name) as well as drawings stored on disk (From File).
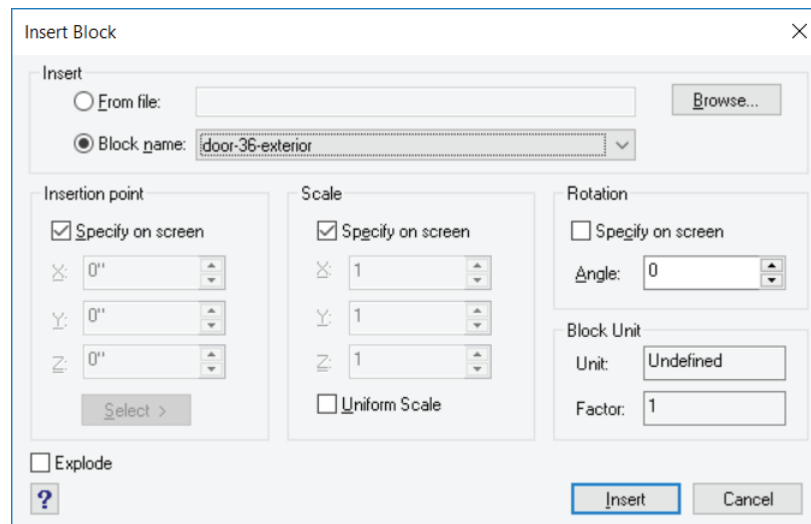
2. Make the following changes to the options:

Block Name	door-36-exterior
Insertion Point	Specify on Screen
Scale	Specify on Screen
Rotation	(turn off Specify on Screen)

    The result should look like the dialog box above.

3. Click **Insert**, and then user the NEArest entity snap to locate the insertion point on the inner wall line.

> Multiple blocks/<Insertion point for block>: **nea**
>
> Snap to nearest of: *(Pick a point on the inner wall line)*
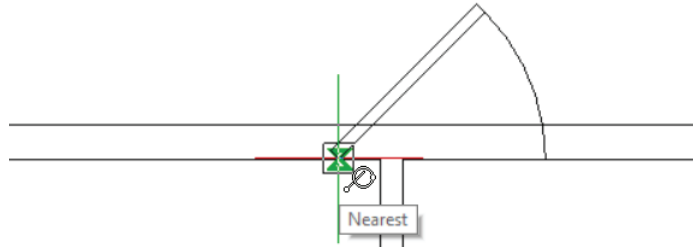
Notice that progeCAD ghosts an image of the block. That allows you to see where the block will end up.

4. The block is not, however, positioned correctly. The block needs to be flipped horizontally and vertically. progeCAD does this when you specify negative scale factors, as follows:

> Corner/XYZ/X scale factor <1.000000>: **y**
>
> Y scale factor: < Equal to X scale (-1.000000)>: **-1**
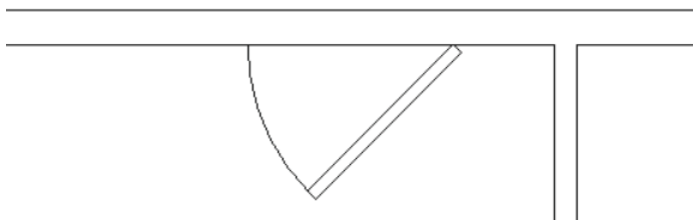
5. Use the **Line** and **Trim** commands to cut out the door opening, and then save the drawing.

Add Window Blocks (Unit Blocks)

The apartment needs windows. The sketch shows their location and length.
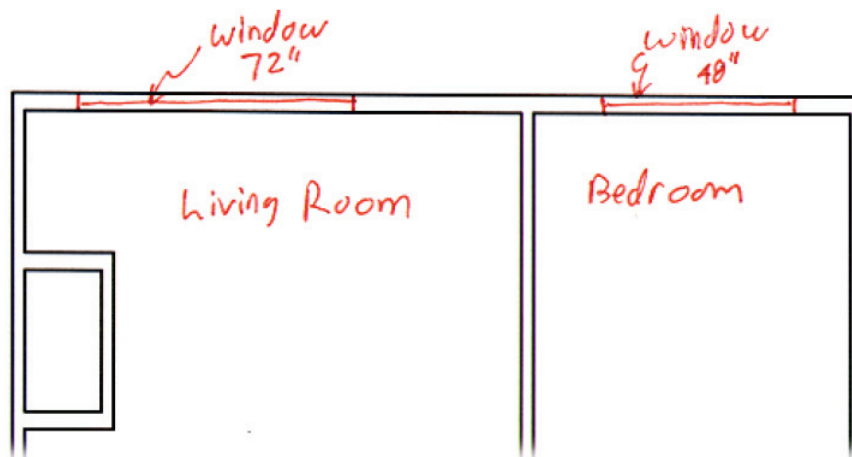
Window symbols typically look like this:

Insertion point

Window pane

6" wide
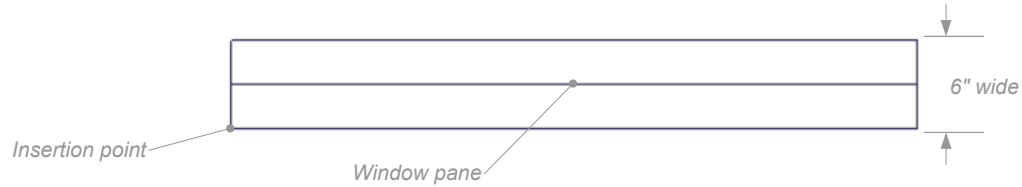
The center line represents the window pane; the other lines are the outline of the window frame. The symbol is 6" tall, because this window fits the 6" exterior walls.

But when you draw the block, you will draw it just 1" wide, because it is a *unit block*. Unlike the door block, whose size does not change, unit blocks change their size upon insertion. For example, the windows you need are 72" and 48" long.

By drawing the block 1" long, the scale factor determines the inserted length. Need a 72" window? Specify 72 as the x-scale factor. Unit blocks make it that easy.

Follow these general steps to create the window block, and then insert it in the drawing:

1.   Start a new drawing.
2.   Draw the 1 x 6 window symbol with the **Line** command.
3.   Convert to a block, naming it "Windows-Exterior." Use the lower left corner as the insertion point.
4.   Save the drawing as "Window-Exterior."
5.   Using Explorer, copy and paste the block to the floor plan drawing.

Using the -Insert Command

This time, use the **-Insert** command to place blocks. This is a third way that progeCAD provides to insert blocks, in which you view and enter options at the command line. The benefit of this version of the Insert command is that it has a Multiple option that lets you insert a block quickly and repeatedly.

1.   Enter the -Insert command:
     Command: **-insert**
2.   Notice that progeCAD is displaying its prompts on the command line. The first one asks you which block to insert, and so provides you four ways to specify the name.
     Specify block to insert or [? to list blocks in drawing/BROWSE/EXPLORE]:

     •   **Specify block to insert** — enter the name of the block, if you remember it. If you don't remember it, then use one of these methods to figure it out:
     •   **? to list blocks** — type '?' and progeCAD lists the names of blocks already in the drawing.
     •   **BROWSE** — type 'browse' and progeCAD displays the Insert Block dialog box, from which you can select any DWG file from the computer's storage or the network

- **EXPLORE** — type 'explore' and progeCAD displays the Explorer palette, from which you visually select a block.

Enter "window-exterior" and then press Enter.

> Specify block to insert or [? to list blocks in drawing/BROWSE/EXPLORE]: **window-exterior**

3. Now comes a prompt that lists all the options available for inserting the block:

> Insertion point for block or [Scale/X/Y/Z/Rotation/Multiple]:

Option	Meaning
Insertion Point	Specify where to place the block in the drawing
Scale	Specify a single scale factor that sizes the block in the x, y, and z directions equally; the scaling is done relative to the insertion point; a negative scale factor makes the block smaller.
X, Y, Z	Specify scale factors separately for the x, y, and z directions
Rotation	Specify the angle at which to rotate the block about the insertion point
Multiple	Repeats the command to insert multiple instances of the block

You are working with a unit-sized window block that needs to be scaled horizontally (in the x direction) only. So enter X to specify the x-direction scale factor:

> Insertion point for block or [Scale/X/Y/Z/Rotation/Multiple]: **x**
> X scale factor for block: **72**

Enter 72 for the scale factor, because you want the window 72" wide.

4. You can ignore the other prompts, so it is time to insert the block using NEArest entity snap to aid you:

> Insertion point for block or [Scale/X/Y/Z/Rotation/Multiple]: **nea**
> Snap to nearest of: *(Pick a point on the inner wall line.)*

Notice that the window appears in the drawing at the correct location and width.

5. Repeat for the bedroom window. The final result should look similar to the figure below.
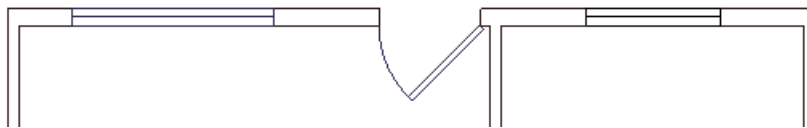
6. Save the drawing.

Accessing Libraries of Blocks

The power behind blocks is that you don't need to create them from scratch. Instead, you can save a lot of time and get drawings completed faster by using libraries of blocks created by others.

Your first source is the free block library provided with copies of progeCAD Professional. To access them, enter the **ICadLib** command, and then notice the dialog box.

Command: **icadlib**
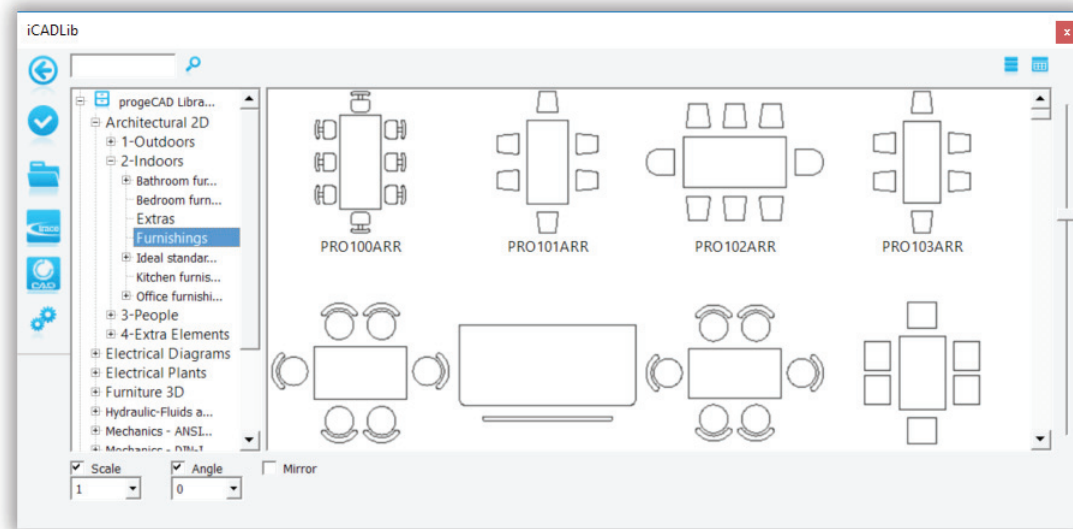
To use the blocks, follow these steps:

1. Go through the tree to find the block you need. Alternatively, enter its name in the search field.
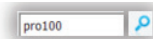
2. Along the bottom of the dialog box, set the insertion parameters: scale factor, angle of rotation, and whether the block should be mirrored.
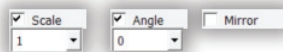
3. Now drag the image of the block into the drawing. At the command prompt, progeCAD asks for the insertion point, as well as gives you another chance to set the scale factor and rotation angle:

    Insertion point for block or [Scale/X/Y/Z/Rotation/Multiple]: *(Pick a point)*

    Remember to use entity snap modes to place the block accurately.

The iCADLib dialog box also provide access to 3D blocks (see figure below) and the mammoth online block libraries maintained by TraceParts and Cadenas.
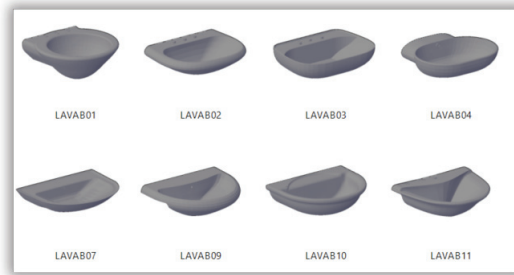
TIP   To add your blocks to the iCADLib block manager, click the **User Folder** button, as described below.

---

iCADLib Toolbar

The iCADLib dialog box has a couple of toolbars. Arranged along the left edge is this toolbar:

Reduce/expand window collapses the dialog box smaller

Insert inserts the block in the drawing

User folder selects folders in which you have stored blocks

Trace Parts goes to the TraceParts Web site

Cadenas goes to the Cadenas Web site

Settings displays the Settings dialog box described later

Along the top of the dialog box is a second toolbar. Left to right, the buttons mean...

...search, list view, and tile view.

TIPS   Any progeCAD *.dwg* drawing file can be used as a block in drawings; it doesn't just need to be drawings of symbols.
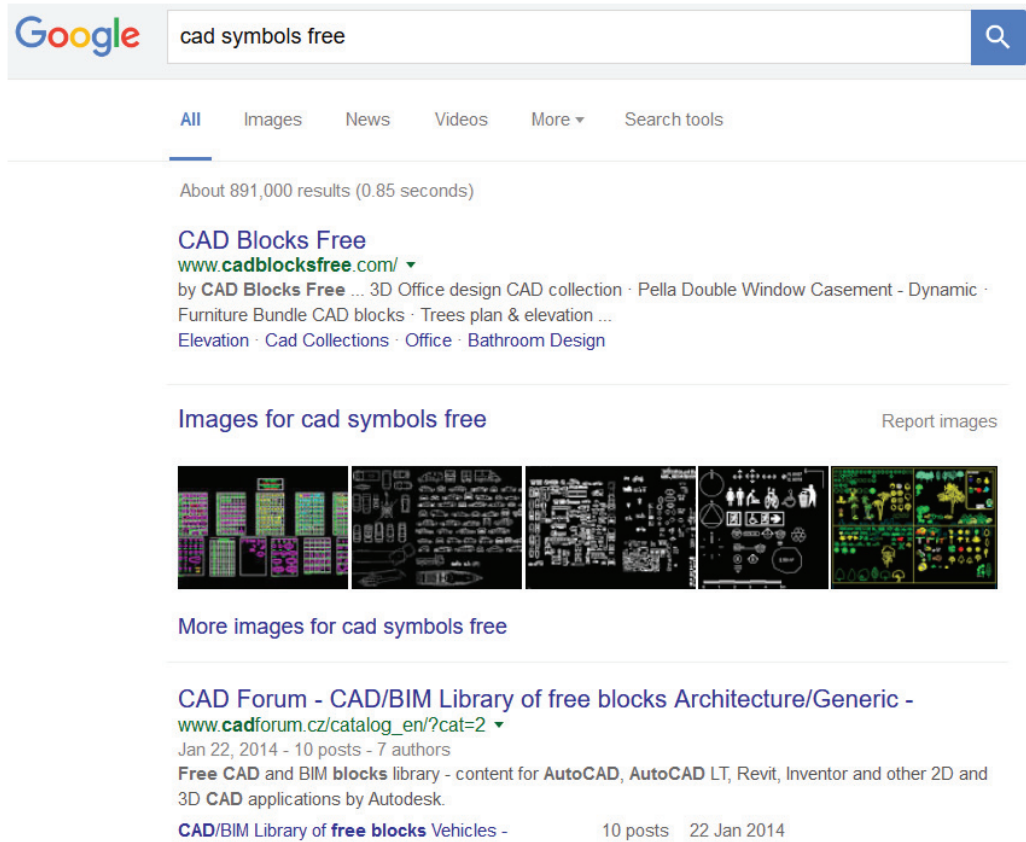
Just as you can open drawings from disc, you can also save blocks to disc. Use the **WBlock** command to save the following entities:
- Selected entities
- A specific block
- Two or more blocks
- The entire drawing

Inside progeCAD 2017

Obtaining Blocks from the Internet

Numerous companies and government agencies provide pre-drawn symbols free through their Web sites on the Internet. Companies often provide symbols free in the hope that you will specify their products, thereby boosting sales, while government agencies provide them free to encourage standardization.

You can find CAD symbols on the Internet by searching the Internet with terms like "CAD symbols free" and "DXF blocks free."

CHAPTER 11

Placing Text

With the drawing of the walls and blocks largely complete, you need to add text to provide explanations of the apartment's features. This chapter describes how to place and edit text. It also shows how to add drawing borders and title blocks.

Important parts of drawings are labeled. Typical labels for floor plans include identification of the rooms and other areas. The figure below illustrates the text you add to the drawing.
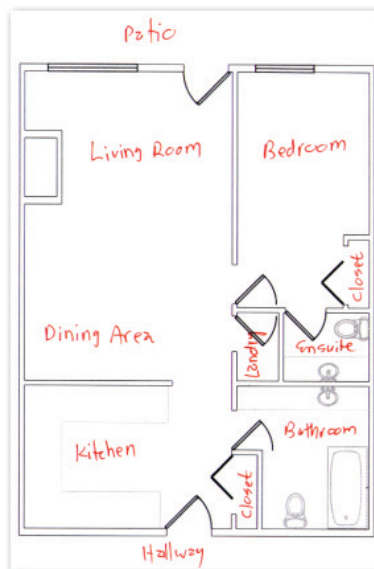
In This Chapter

- **Text** command
- **MText** command
- **MSpace** command
- **Style** command
- **DdEdit** command
- **PSpace** command
- **DText** command
- **MView** command

Places lines of text.

Command: **text**

Aliases: **-t, tx**

Menu: **Insert | Text**

Toolbar: **Draw 2D| Text**

MText places paragraphs of text.

DText repeats the 'Text:' prompt.

Leader places text with leader lines and arrows.

Placing Text (Text)

Begin by labeling the living room with the **Text** command, as follows:

1.   First, though, use the **Layer** command to create a new layer named "Text." All of the text you add to this drawing will be placed on this layer, so ensure the layer is current.

2.   Start the **Text** command.

Command: **text**

3.   Don't let the command's many options concern you. For now, just click anywhere along the left edge of the living room area.

Text: Specify start point or [Style/Align/Fit/Center/Middle/Right/Justify]: *(Pick a point)*

4.   Next, progeCAD wants to know the height of the text.

Height of text <0">: **12**

TIP   Q: How tall should text be? A: Tall enough to be legible when plotted.

The standard for text in drawings is 1/4", while headings are a bit taller at 3/8". If you were to use those numbers for text in this drawing, however, the text would be too small.

Instead, you need to make the text bigger, so that it *appears* 1/4" tall when printed out. How much bigger?

Recall that the **Print** command's dialog box reports the scale factor (see Chapter 9). progeCAD scales the drawing smaller so that it fits the page. For this drawing, the scale factor reported by the Print command is 1" = 4' or 48x smaller.

For 1/4"-tall text to be legible, it has to be 48x larger, or 48 x 0.25" = 12" tall in the drawing.

---

5.   Text can be placed at any angle. This text is horizontal, 0 degrees.

Rotation angle of text <0>: *(Press **Enter** to accept default value)*

6.   Finally, you get to enter the text:

Text: **Living Room**
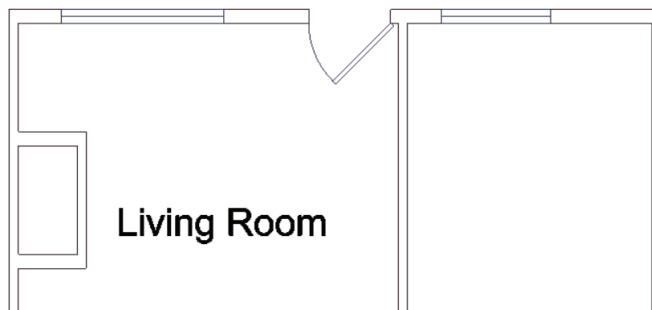
progeCAD places the text. By default, it uses the Arial font, which is also the default for Windows. If you want, you can use a different font. progeCAD works with any font that works with Windows. These are called "TrueType" fonts, and have a file extension of *.ttf*. As well, progeCAD can use any font that come with AutoCAD; these have the file extension of *.shx*.

Creating Styles (Style)

To use a different font in drawings, you use the **Style** command.

You may have noticed that the Text command lacked a "Font" option for specifying the font name. That's because fonts are specified by the Style command. You may be familiar with *styles* in documents: styles define fonts and their properties for headings, paragraphs, text — bold, underlined, superscript, and so on. progeCAD uses styles in the same way to define the look of text.

To change the font used for the "Living Room" text, follow these steps:

1. In the command bar, enter **style**:

   Command: **style**

   Notice that progeCAD displays the Text Style dialog box.
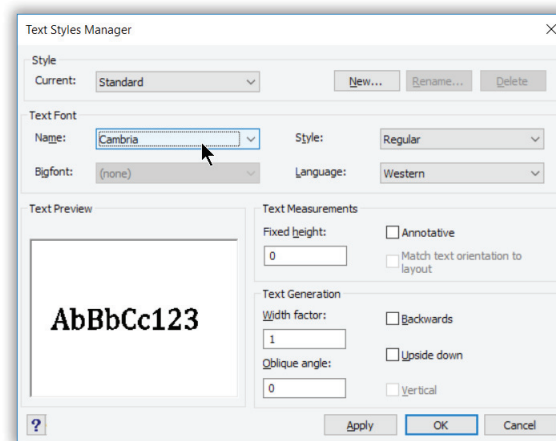
2. In the Text Font area, click the **Name** droplist.
3. Select "Cambria," a modern serif font that is available on most newer computers. (If it is not available on your computer, then choose another font name.)
4. Click **OK**. Notice that the "Living Room" text instantly changes to Arial font.
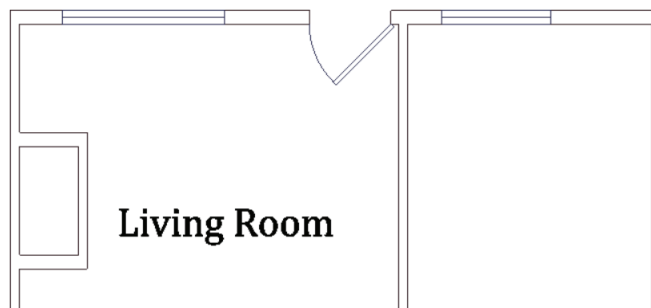
Controlling Text Styles

progeCAD provides three ways to create and change text styles:

- **Style** command displays a dialog box.

- **-Style** command prompts in the command bar; it is meant primarily for use in macros and LISP.)

- **ExpStyles** command displays the Explorer-Text Styles window.

All three do the same job; they just have different user interfaces. Try each out to see which you prefer. Here is what the Explorer-Text Styles user interface looks like:
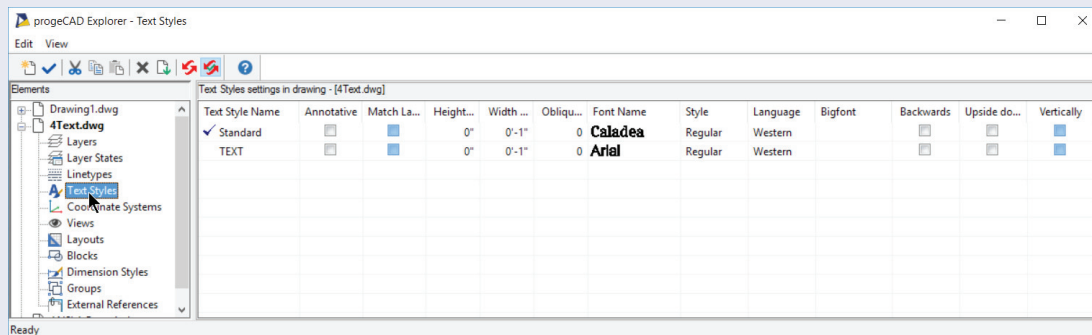
The options have the following meaning:

Annotative turns on annotative scaling for the text.

Match Layout Orientation rotates text when the layout is rotated.

Height specifies the height of the text. When set to 0, the Text command prompts you for a height.

Width Factor makes characters wider (values over 1.0) or narrower (less than 1.0). Using a value of 0.85 allows you to squeeze in 15% more text per line, while keeping text legible.

Oblique Angle slants the characters forward (positive angles) or backward (negative). The old way of creating italic-looking text before italics became common in fonts.

Font name selects the font file from those available on your computer.

Style sets the text as regular, italic, bold, or bold-italic.

Language specifies the human language version of the font file to use.

Backwards draws text backwards; used for printed circuit board drawings.

Upside down same as specifying 180 degrees for the rotation angle.

Bigfont used for *.shx* fonts with Chinese and Japanese characters.

Vertically draws text vertically, one character below the next.

progeCAD can place text at any height and at any angle. The label for the closet, for example, is at right angles. Repeat the Text command, and then specify a rotation angle of 90 degrees, as follows:

Command: **text**

Text: Specify start point or [Style/Align/Fit/Center/Middle/Right/Justify]: *(Pick inside the closet)*

Height of text <1'-0">: *(Press Enter to accept default height)*

Rotation angle of text <0>: **90**

Text: **Closet**

TIP Text is just like other entities: you can use the **Move** and **Copy** commands to reposition and copy text. But not all editing commands work on text; for example, you cannot use the **Stretch** or **Explode** commands.

<div style="border:1px solid #888; padding:1em;">

DTEXT COMMAND

Places lines of text, repeating the 'Text:' prompt.

Command: **dtext**

Alias: **dt**

ALTERNATIVES

MText places paragraphs of text.

Text places lines of text.

Leader places text with leader lines and arrows.

</div>

Label Drawings (DText)

Repeatedly responding with Enter to options whose values don't change — such as height and rotation angle — gets annoying after a while, and so progeCAD has a second text command, named **DText** (short for "dynamic text"). It asks for options once, and then lets you repeatedly pick insertion points for placing text. This command is excellent for placing labels all over drawings, and now you use it to finish placing text in the drawing.

1.   To place the remaining room labels, enter the **DText** command in the command bar:

Command: **dtext**

2.   The prompts that follow are identical to that of the **Text** command:

Text: Specify start point or [Style/Align/Fit/Center/Middle/Right/Justify]: *(Pick a point in the drawing)*

Height of text <1'-0">: *(Press **Enter**)*

Rotation angle of text <0>: *(Press **Enter**)*

Text: *(Enter a room label, such as **Kitchen**, and then press **Enter**)*

Notice that the text appears, letter by letter, as you type.

3.   When you press **Enter** after 'Kitchen,' the command does not end. Instead, the 'Text:' prompt repeats itself.

4. Position the cursor in another room, and then click. This indicates the new insertion point from which the text starts. Enter another room label.

    Text: *(Move the cursor, and then click. Enter another room label, such as **Bedroom**)*

5. Finish labeling the floor plan: add text to the dining room, and so on.

6. When done, press **Enter** at the 'Text:' prompt to exit the command.

    Text: *(Press **Enter**)*

When done, the drawing should look similar to the figure below. Save your drawing.
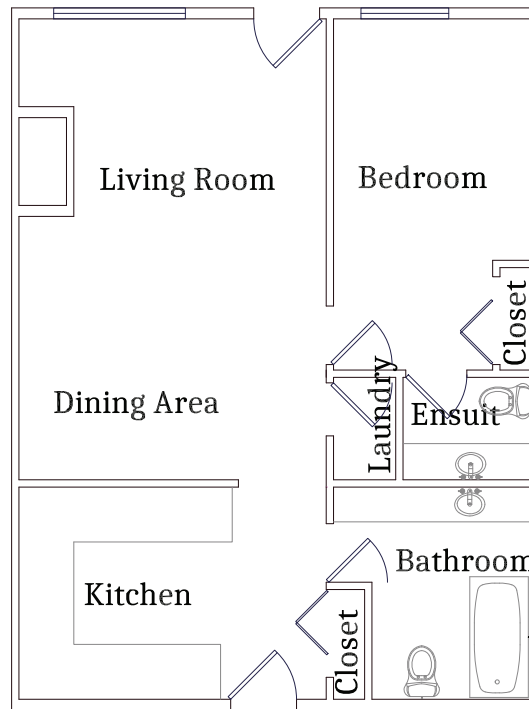
Editing Text (DdEdit)

Did you notice that "Ensuit" was misspelled? The **DdEdit** command allows you to change the wording of text, and correct mistakes. (DdEdit is short for "dynamic dialog editor.") To use it, you just double-click words to change them.

TIP   progeCAD spell checks any kind of text in drawings, including single-line text, multiline text, text in block attributes, and in external references. To run the spell checker, enter the **Spell** command.

---

1. To edit the word, select it, and then double-click. Notice the Text dialog box.
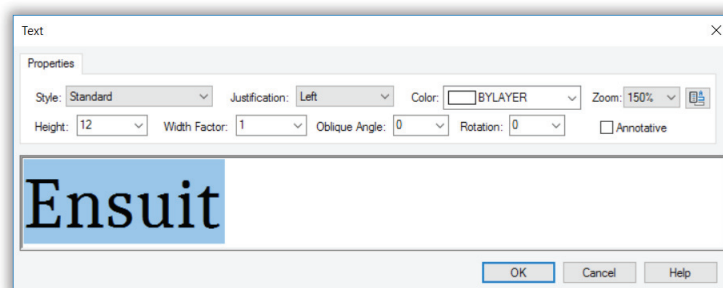
Understanding Text Justification Modes

If you have difficulty placing text, it could be a problem with *insertion points* or *justification*.

TEXT INSERTION POINTS

Text entities have insertion points just like blocks. For text, the insertion point is where the text starts. To see the insertion point of a piece of text, select it: the grip you see is located exactly at the insertion point. (You can use the **INSertion** entity snap with text.)

The insertion point can move around, depending on the *justification*. Justification tells pro-geCAD where to start the text. There are 15 justification modes, of which these three are used the most often:

Left — the default, a.k.a. "Start point"

Right — for right-justified text

Center — for center-justified text

The others are Align, Fit, Middle, TL, TC, TR, ML, MC, MR, BL, BC, and BR. Two sound the same: **Align** lets text keep its width factor (but changes the height to fit), while **Fit** lets text keep its height (but adjust the width factor to fit).
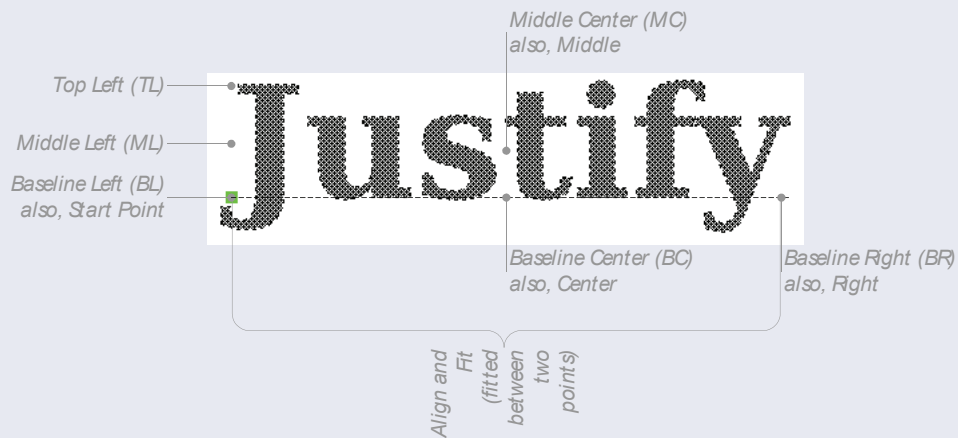
2. In the text box, change Ensuit to "Ensuite."

3. Click **OK**. Notice that the word is corrected in the drawing.
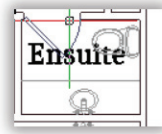
4. Press **Esc** to end the command.

TIP   An alternative method for changing the spelling of words and other properties
select the text, and then use the Properties palette. If necessary, enter the **Properties**
command to open the palette.
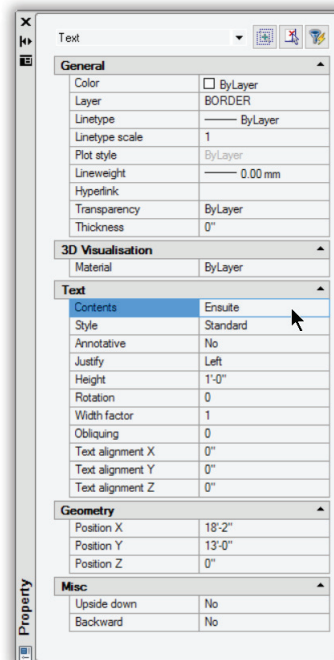
Borders and Title Blocks

Drawings are often drawn with *borders*. Borders are useful for identifying the purpose of drawing, and other info, such as who made the drawing and when. If nothing else, borders give drawings a professional finished look.

Engineering and architectural firms usually have their own drawing border standards; in some cases, clients require the use of their specific borders. In addition, standards organizations such as ANSI in the United States, DIN in Germany, ISO internationally, and JIS in Japan specify the design of borders.

There are different borders for different sizes of papers, ranging from A- to E-size in imperial measurements and the equivalent A4- through A0-sizes in metric jurisdictions. The figure

below illustrates the A-size portrait border provided by progeCAD.
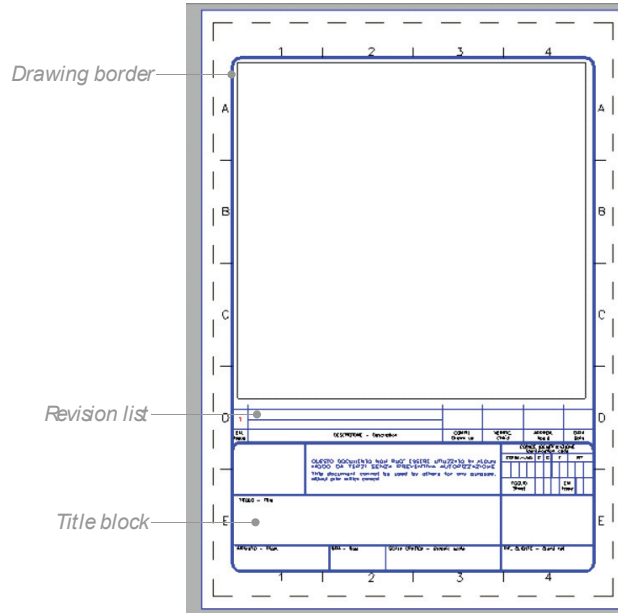
Drawing Border shows the limits of the drawing.

Revision List provides a record of changes made to the drawing.

Title Block records information about the drawing, such as its title, the date it was drawn, and the name of the drafter.

TIP   progeCAD comes with a dozen templates that have title blocks embedded. To access them, enter the **PNew** command, and then click on the **Use A Template** button.
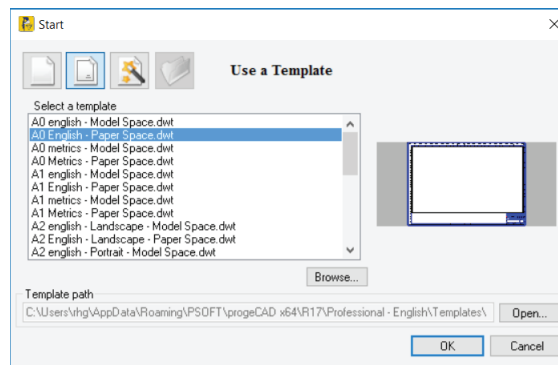
Borders are available in a range of standard sizes. The smallest is called "A", which you may recognize as the 8-1/2 x 11" letter-size paper used in many offices. Each larger size doubles in area:

ANSI Size	Imperial Dimensions
A	8.5 x 11" (landscape) and 11 x 8.5" (portrait)
B	11 x 17" (landscape only)
C	17 x 22"
D	22 x 34"
E	34 x 44"

The equivalents in metric ranges from A4 (like A) through A0 (like E).

Layout Drawings (MView)

You could draw the border around the drawing, but that leads to scale issues. As with text, the border has to be scaled to fit the size of paper. Unlike text, the border will usually change according to the size of paper on which the drawing is printed.

Until now, you have been drawing in "model" mode, but progeCAD provides a second way of looking at drawings, called "layout mode." At the bottom of the drawing area, notice that there are three tabs: Model, Layout1, and Layout2.
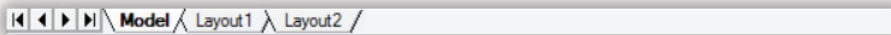
As the names suggest, Model is for creating and editing the model (drawing), and Layout mode is for laying out the drawing in preparation for plotting. The number "1" next to Layout ("Layout1") is indicates to you that each model can have one or more layouts. (Layout is a newer name for what used to be known as "paper space.")

You create layouts for different sizes of drawing and types of plotters. For example, you might want a layout for plotting to an A-size laser printer in monochrome, and then another layout for plotting to an E-size inkjet plotter in color.

Here's how to use layouts:

1. Switch to layout mode by clicking the **Layout1** tab.
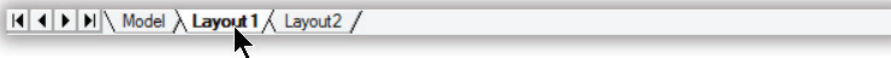
    Notice that the drawing area changes its look, showing the floorplan inside a dashed rectangle.
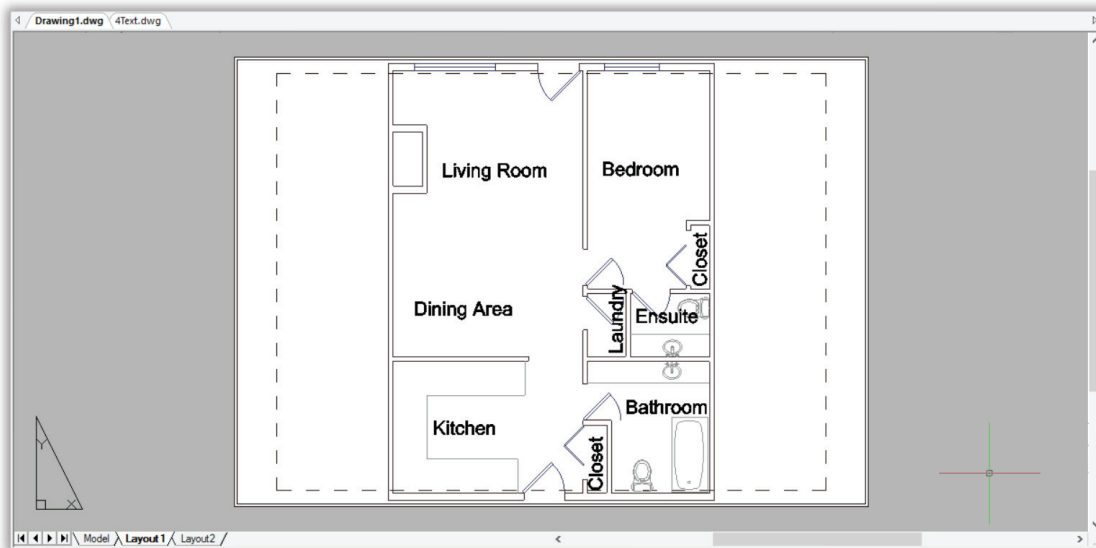
2. Draw a border and title block using the dimensions illustrated below. Draw as much or as little of the border as you wish.
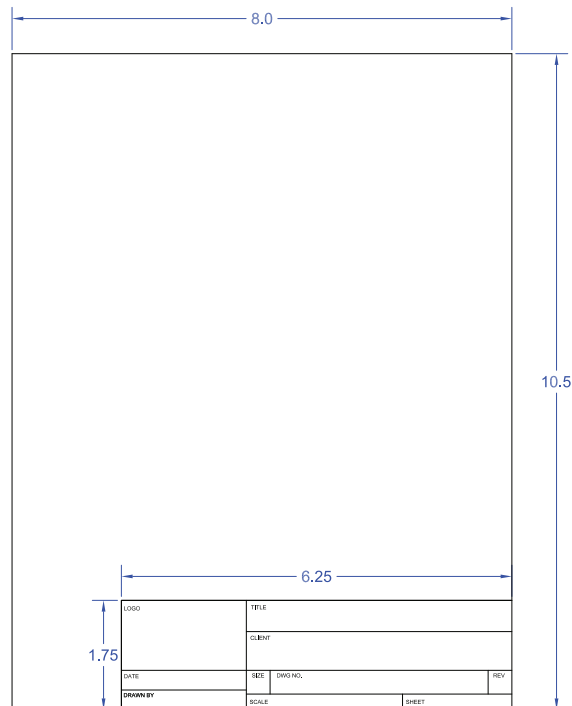
Or, use the **Insert** command to place the *ANSI A Portrait.dwg* file as a block. Use the default settings: insertion point 0,0; scale factors = 1; rotation = 0.

You are looking at two "overlapping" drawings. One is the floor plan in *model* space; the other is the border and title block in *paper* space (a.k.a. layout). The two spaces are independent of each other, yet linked by a viewport.
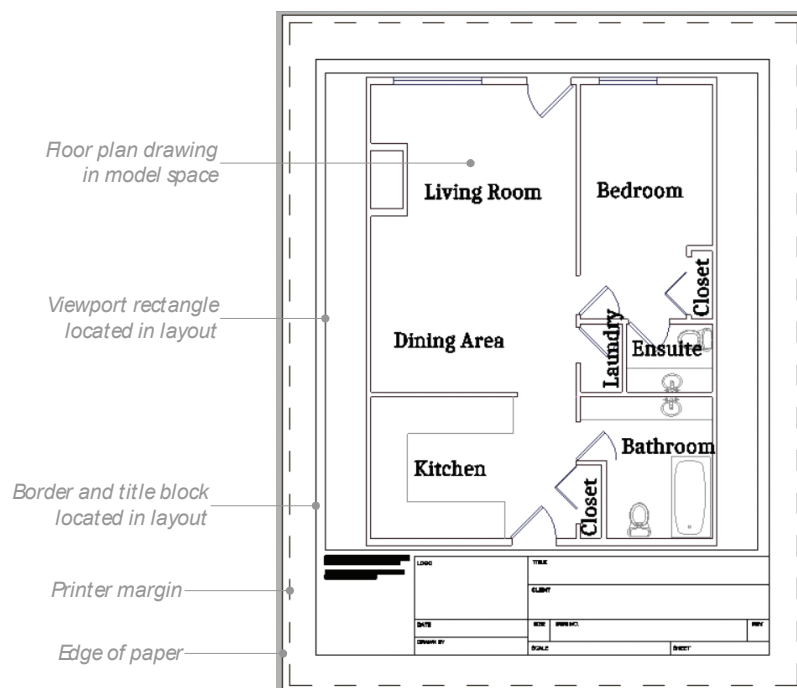
While you can see entities from model space in the layout, the reverse is not true: entities drawn in layouts are not seen in model space. When you click on the Model tab, you no longer see the border. You learn more about these particulars in the following section.

Paper Space Exercises (MSpace, PSpace)

Paper space (layout mode) can be confusing to new CAD users, so here are some exercises to help you understand the differences. Be sure the drawing is in layout mode by clicking the Layout1 tab.

1. Move the cursor over any part of the floor plan and then click. Notice that progeCAD doesn't react; indeed, it cannot find any entities.

   Lesson: When in layout mode, model space entities are inaccessible. You can see, but you cannot touch.

2. Select a part of the border or title block. Notice that you can select it.

   Lesson: When in layout mode, paper space entities can be selected and edited.

3. Select the viewport rectangle. Notice that square grips appear. Drag one of the grips so that the viewport changes its size. (See figure below.)

   Lesson: Viewports are editable entities. You can move, copy, stretch, erase them, and place them on layers — as well as apply properties, such as color, linetype, and lineweight.
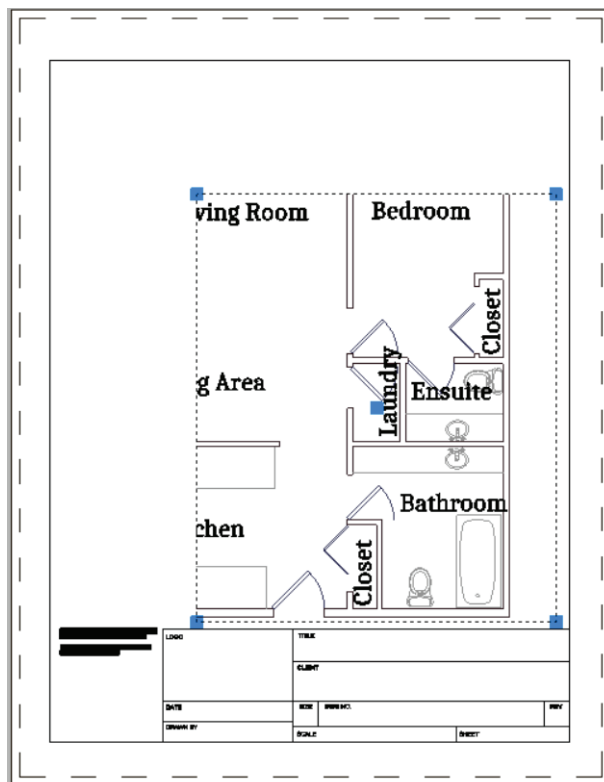
4. Press **Esc** to deselect the grips, and then use the **U** command to revert the viewport to its original shape.

5. Use the **la** alias to freeze the "Viewport" layer. Notice that the viewport rectangle disappears, but that the floor plan drawing remains.
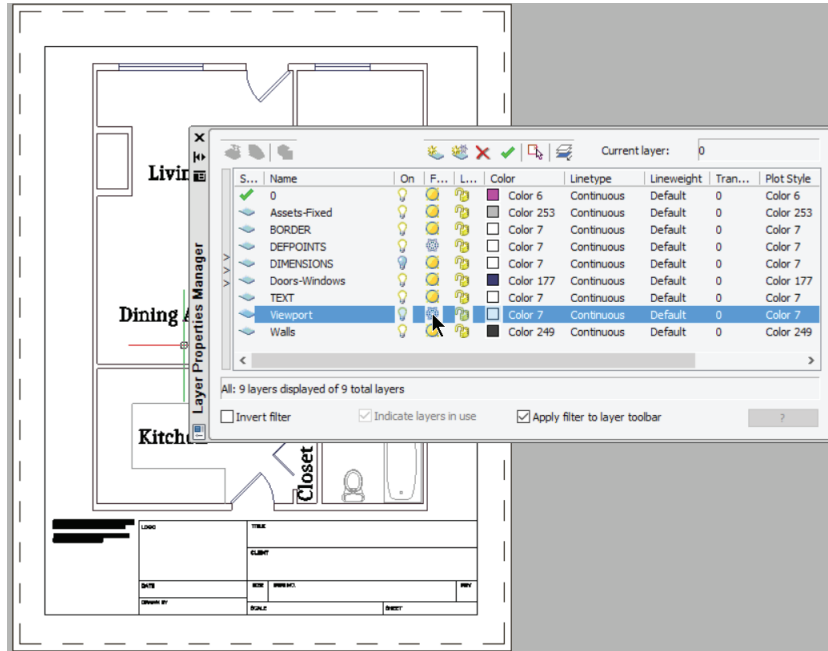
Lesson: The viewport's properties are independent of the viewport content (the model space drawing).

6. Use the **Zoom Window** command to zoom into the title block. Notice that the floor plan model zooms along. Use the **Zoom Extents** command to revert the view.
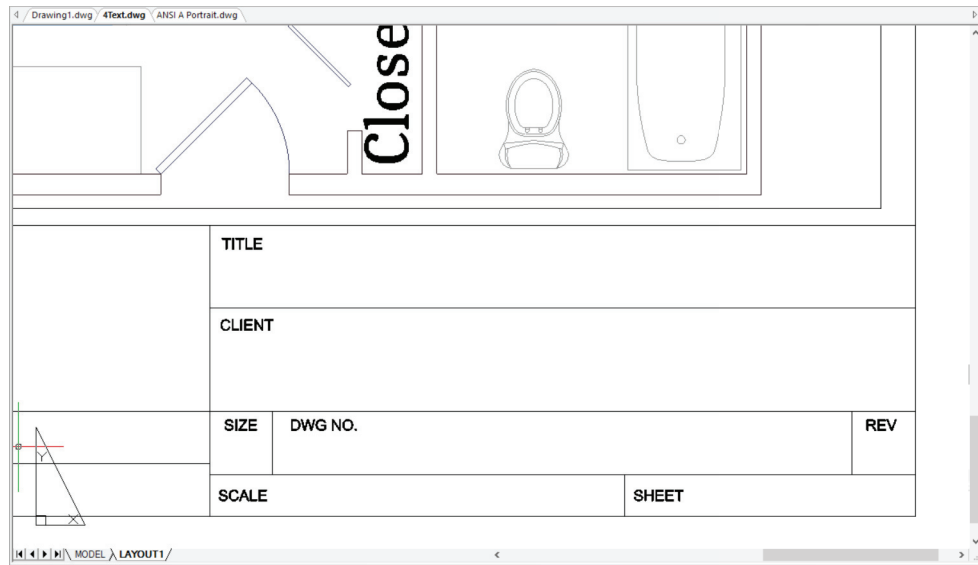
Lesson: You can zoom and pan in layout mode just as in model mode.

7. A button on the status bar lets you switch between the two spaces, model and paper.
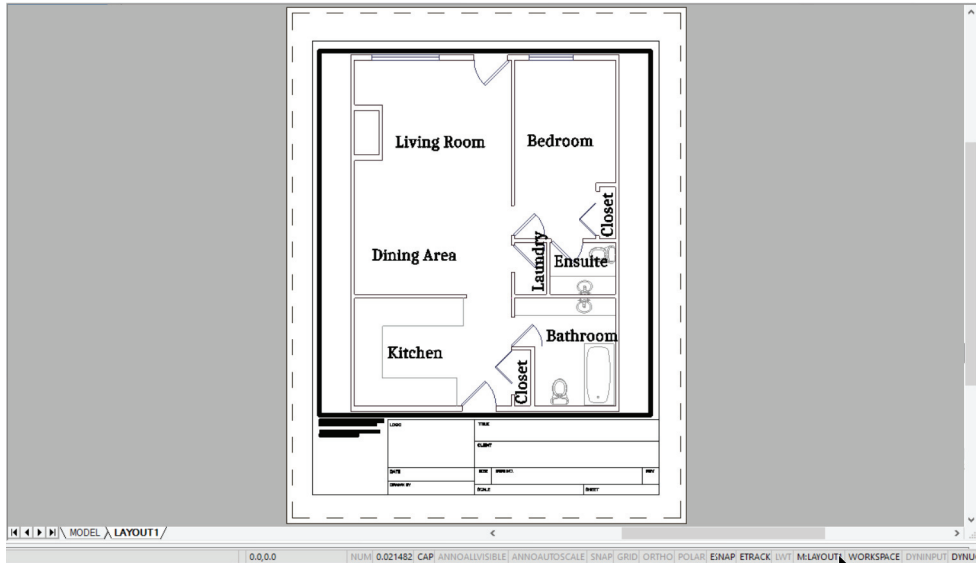
Click **PAPER**; notice that it changes to **M:LAYOUT1**. (The action executes the **MSpace** command.) This means that the drawing is in model space of Layout1.

Notice too that the viewport now has a heavy border. This is a visual clue that you are now working in model space, and is especially useful when a layout has more than one viewport. To move the focus to another viewport, press **Ctrl+R**.

> **TIP**   As an alternative clicking the button on the status bar, you can double-click inside a viewport to access model space. To get back to layout mode, double-click outside the viewport.

8. Repeat the actions you used above: go into model space by double-clicking inside the viewport, and then...

   a. Select an entity in the floor plan: this time it is highlighted, because it is in model space.
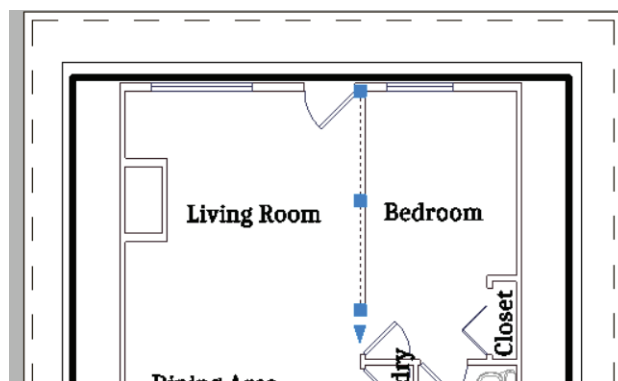
   Press **Esc** to cancel grip editing mode.

   b. Select the border: this time progeCAD cannot find it, because the border is in paper space.

c. Zoom into a part of the floor plan: this time the floor plan (in model space) becomes larger, but the border and title block remain unchanged (in paper space).
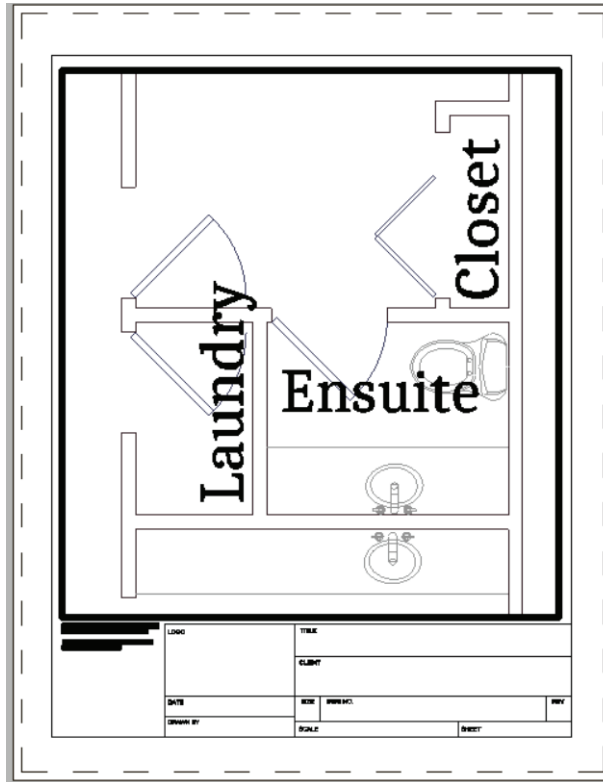
d. Use the **Zoom Extents** command to return the view to normal.

9. Click **M:LAYOUT1** on the status bar to return to paper space. (This executes the PSpace command.)

Place Paragraphs of Text (MText)

After learning about the Text and DText commands, you might be surprised to learn that progeCAD hosts yet another command for placing text. The **MText** (short for "multi-line text") command is designed for placing entire paragraphs of text, such as notes. But it is especially useful for text that must contain a variety of text effects, such as multiple fonts, boldface, superscripted characters, and several colors.

1. To place notes in the drawing, start the **MText** command.
Command: **mtext**

MTEXT COMMAND

Places paragraphs of text.
Command: **mtext**
Aliases: **t, mt**
Menu: **Insert | Multiline Text**
Toolbar: **Draw 2D | Multi-line Text**

ALTERNATIVES

Text and **DText** place lines of text in drawings.

---

2. progeCAD prompts you to pick two points. These form the rectangle within which the paragraphs of text are fitted. The height of the rectangle is unimportant, as progeCAD will automatically extend the bottom of the block; the top and sides constrain text.

> Multiline Text: First corner for block of text: *(Pick a point)*
>
> Opposite corner for block of text or [Justification/Line spacing/Rotation/Style/Height/Direction/Width/Columns]: *(Pick another point)*
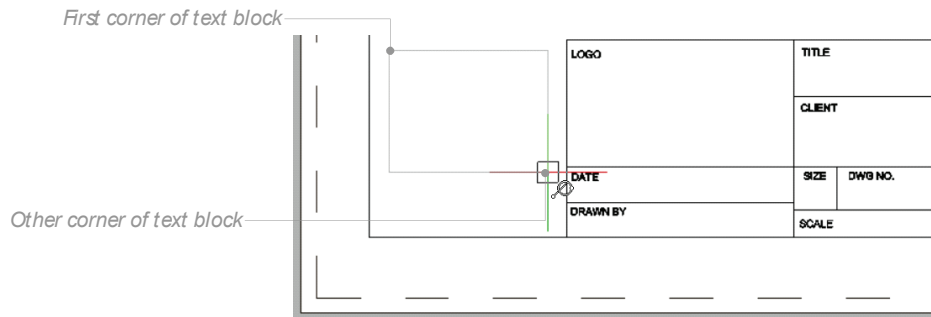
3. After you pick the second point, progeCAD displays the Multiline Text dialog box. It is here that you enter text and format it.
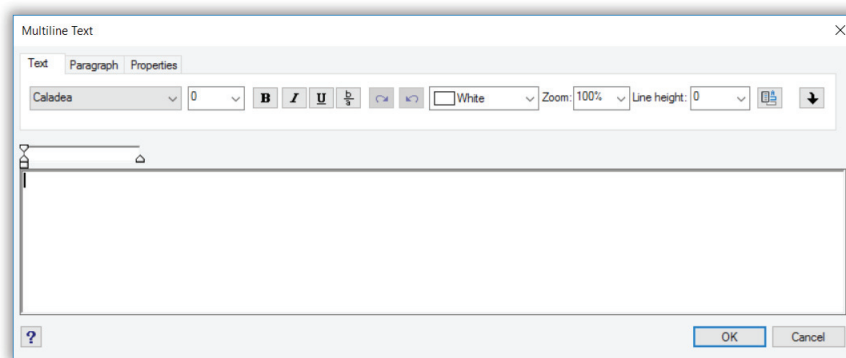
4. Enter text, such as the notes illustrated by the figure below.

TIPS   To format a selection of mtext, highlight it (drag the cursor over the text), and then select the formatting option.

You can copy text from another source, such as a Word document, and then use **Ctrl+V** to paste the text in the mtext editor.

After mtext is placed in the drawing, you can edit it by double-click the mtext; in the

Entity Properties dialog box, select **Full Editor**.

---

5. Click **OK** to dismiss the dialog box, and so place the text in the drawing.

6. Enter the **Preview** command, first in layout, and then in model. You see a different version of the drawing, depending on the mode in which you enter preview:

> **PAPER** – in paper space, both model and paper space (floor plan and border) are printed
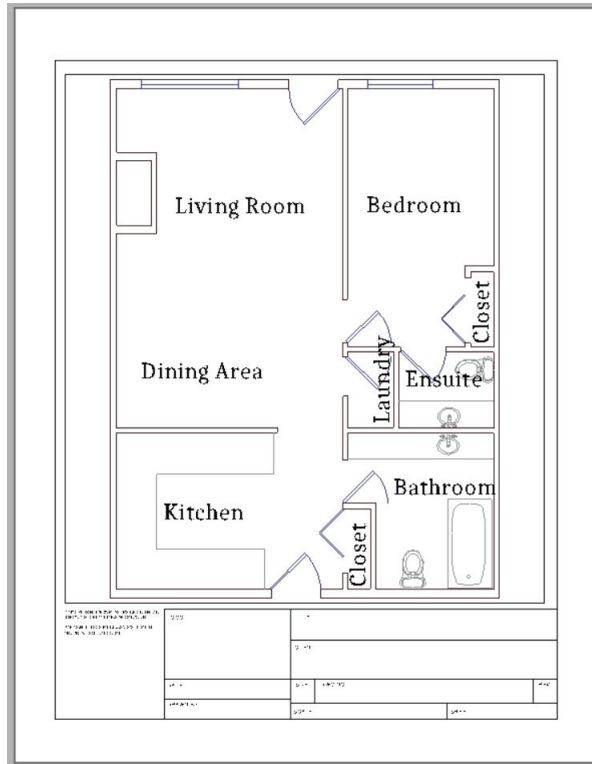
> **MODEL** — in model space, only the model (floor plan) is printed
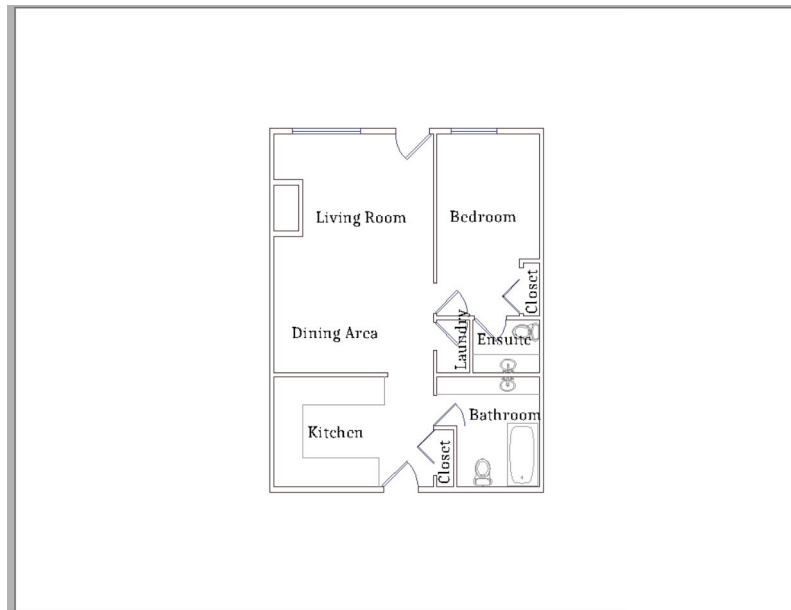
7. Print the drawing, if you wish.
8. Save your work with the **QSave** command.

Understanding the MText Dialog Box

Here is the meaning of the various buttons and droplists in the MText dialog box. Before changing the format of mtext, remember to select it first. If you forget to highlight the text, MText won't make the changes.

TEXT TAB
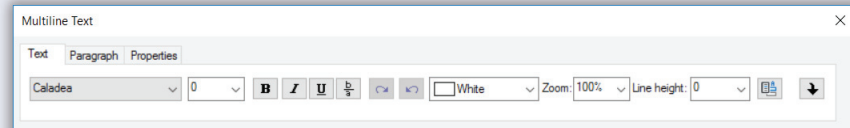
Font droplist select fonts. You can chose from any TrueType and AutoCAD font installed on your computer.

Height droplist specifies the height of the text in inches. The droplist contains the default height of 0.2 units, as well as any recently-used heights, and you can type in any other value.

Bold, **Italic**, **Underline** add bolding, italics, and underlining to text.

b/a stacks selected text that's on either side of a slash.

Undo, Redo undoes and redoes the last action(s) in the dialog box.

Color droplist changes the color of the selected text.

Zoom droplist enlarges and reduces the size of the text in the dialog box; does not affect the size of the multiline text placed in the drawing.

Line Height droplist changes the spacing between lines, such as '2' for double spacing.

Insert/Edit Field displays the Field dialog box for creating and editing updatable field text.

More displays a menu of additional actions; see next page.

PARAGRAPH TAB
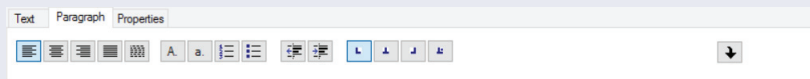
Left, Center, Right, Full, Adjust justification. The difference between full and adjust is that the former adds spaces between words while the latter adds spacing between characters.

Letter, Number, Bullet adds sequential letters, numbers, or bullets to paragraphs.

Outdent, Indent moves the left edge of the text in or out.

Left, Center, Right, Decimal justification for tabs. Enter values in the tab field, below the toolbar:

PROPERTIES TAB
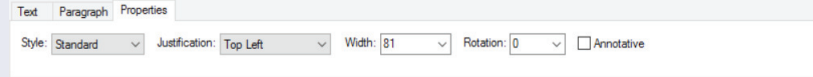
Style droplist selects a text style defined by the Style command. The properties set by the Text tab override the properties specified by the style.

Justification droplist specifies how the paragraphs of text fit within the mtext rectangle.

Width changes the width of the text block; enter another width, or select **No Wrap** (text is not constrained to the mtext rectangle).

Rotation rotates the entire block of text; select one of the preset angles, or enter a specific angle.

Annotative toggles annotative scaling for the text.

MORE MENU

The More menu repeats some of the functions found in the toolbars, but also has some unique ones worth mentioning.
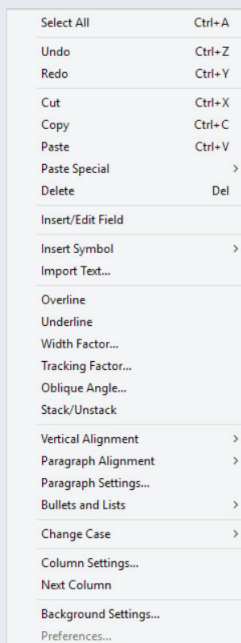
Insert Symbol is an easy way to add common symbols, like degree, angle, and Omega.

Overline through **Oblique Angle** mimic the settings in the Style dialog box.

Change Case changes the case of selected text to uppercase or low-ercase.

Column Settings splits paragraphs into adjacent columns.
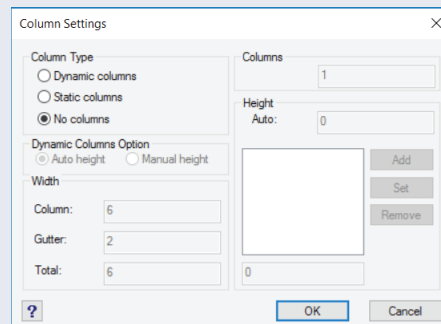
Background Settings lets you apply a colored rectangle behind the text.
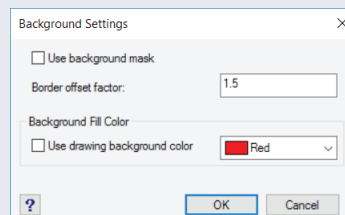
Annotatively-Scaled Text

You needed to work out in this chapter the scale factor for text. The height of text takes into account the size of paper on which the drawing will be printed. If the paper size never changes, then all is fine.

If paper sizes change, such as sometimes printing on A-size and sometimes on E-size paper, then you need a way to change the size of text easily. progeCAD provides this through *annotative scaling*, which selectively resizes text, as well as other annotative entities such as dimensions, hatch patterns, and linetypes.

After you turn on the annotative scaling option, progeCAD determines the correct height to use for text (as well as the correct scale factor for hatch patterns, and so on). Indeed, it can store several annotative scales per entity, hiding the scales that are not needed for the current situation.

You can think of annotative scaling as a "master scale factor" that affects only annotative entities in three areas of a drawing: with annotative entities, in viewports, and on plots. To understand this better, I wrote this rule:

<p align="center">Annotative scale = Viewport scale = Plot scale</p>

The scale at which you plot the drawing determines the scale factor of annotative entities. (If your drawing uses viewports, the same scale applies to them.)

To use annotative scaling, you turn it on in styles, whether for text, dimensions, or hatch patterns: just click the **Annotative** check box. The figure below shows the Annotative property in text styles. (For dimension styles, look for the Annotative check box in the Fit tab.)
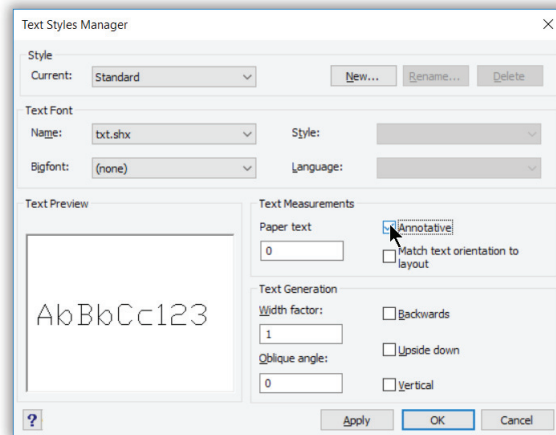
Linetypes are different in that they use the **MsLtScale** variable to match the current annotative scale factor. As a side note, annotative scales apply to entities, not to layers.

You can't type in an annotative scale factor; you use the ones that are provided by the **ScaleL-istEdit** command, which lets you add and remove scale factors from a list. progeCAD comes with a long list of them already in place — 33 of them, some of which are shown below.
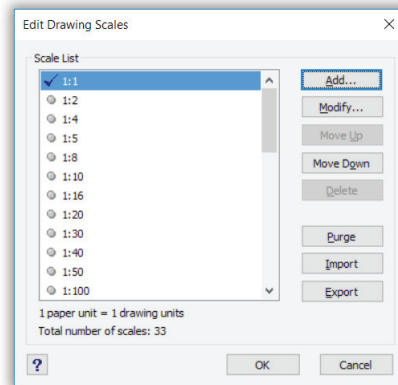
Using Annotative Scaling

Let's see how annotative scaling works for real by applying a scale factor to the floor plan in layout mode. When a drawing is first opened in a viewport, it is most likely displayed at a non-standard scale, because progeCAD automatically fits the model by its extents to the area inside the viewport.

1. Make sure the drawing is in layout mode by clicking the **Layout1** tab.
2. Double-click inside the viewport to enter model space. You know you are successful when the viewport border becomes a thick rectangle.
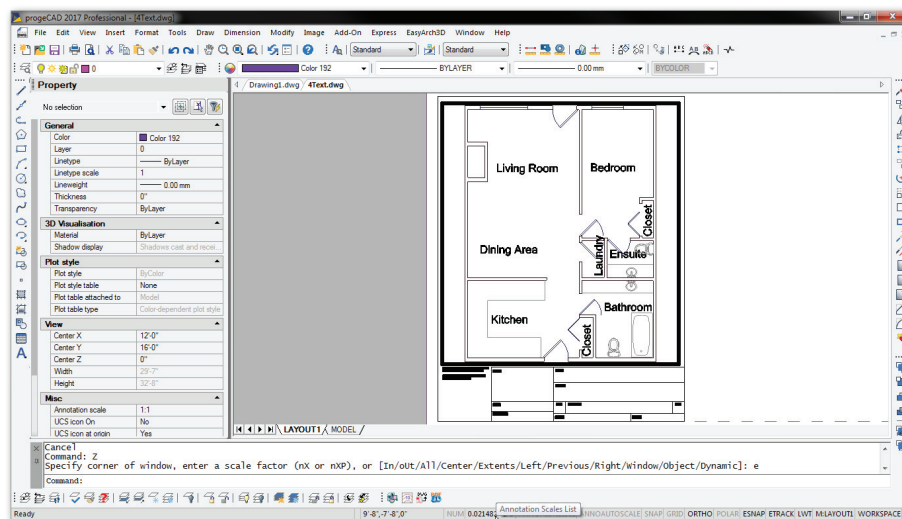
3. Look at the status bar for the annotations scale. In my version of the drawing, it has an initial scale of 0.021482 — or 1:46.5506 expressed as a scale factor. The annotations scale may be slightly different with your version of the drawing.

> **TIP**   If you do not see the annotative scale button on the status bar, right-click in the 'Ready' field, and then choose **Annotations Scale List** from the shortcut menu.

4. Because 1:46.5506 is a non-standard scale factor, you have to change it to meet these two criteria:

   • The entire drawing must fit the viewport and somewhat smaller than the viewport

   • The drawing must be at a standard scale

TIP To convert a scale into the scale factor, divide 1 by the scale. In this case...
$$1 / 0.021482 = 1:46.5506$$

You change the oddball scale to a standard one through the Annotation Scale List button on the status bar or in the Properties palette. On the status bar, the annotative scale button is located as shown in the figure above.

For the scale factor, pick the next largest whole number to 1:46.5506, because standard scale factors are always whole numbers, such as 1:50. Choosing the next largest number makes the drawing slightly smaller in the viewport. You don't want some of the drawing hidden by the edge of the viewport!

To pick the annotative scale factor, right-click the **Annotations Scale List** button, and then select **1:50**.
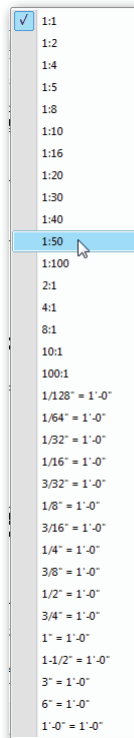
You know that you have picked the correct scale when the drawing becomes slightly smaller, but not too much smaller. Notice also that the reported scale matches the annotation factor you selected. From now on, the viewport scale is locked to the annotation scale. This means that any change you make to the annotation scale changes the viewport scale automatically.

5. With the annotative scale factor set up, test the annotation scaling feature by placing two kinds of text in Model tab, *standard* and *annotative*. This lets you see the difference between the two.

First, draw some standard text:

a.   Click the **Model** tab.

TIP   Why place the annotative text in model tab? (Why not in the layout tab?) My CAD-expert friend Bill Fane explains:
"If you zoom and pan while entering annotative text in a layout tab, you mess up the viewport's scale; when you go back into model space, the annotations are missing, because their scale no longer matches the scale of the viewport. For this reason, it is best to enter annotative text in Model tab."

---

b.   Enter the **Text** command to place regular text with the "Standard" style (which is non-annotative) at a height of 1/8" (0.125"):

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point)
Height of text <0.2000>: 0.125
Rotation angle of text <0>: (Press Enter)
Text: Hallway
Text: (Press Enter)
```

Notice that the text looks very tiny. It is so tiny that in the figure I needed to highlight it with a blue grip. It is so tiny because it is placed at the size we normally use for text in drawings, 1/8". The only reason we could see the text earlier in the chapter is because you had manually scaled it up by 48x, and then drew it 6" high!
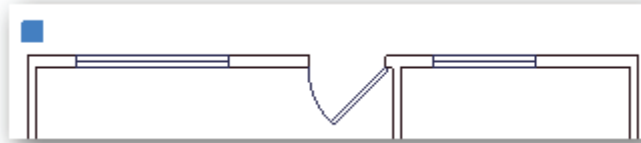
6. If the drawing does not contain an annotative text style, then create one like this:
   a.   Enter the **Style** command.
   b.   Make a copy of an existing style by clicking the **New** button.
   c.   Name the style "Annotative."
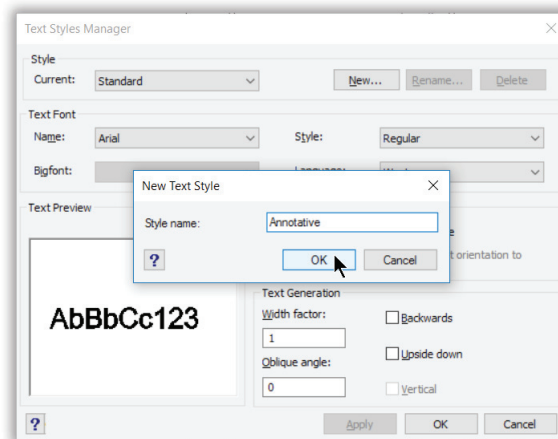
d. Click the **Annotative** option. It's that simple!
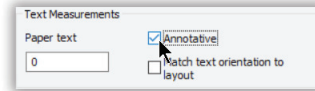
e. Click **OK** to exit the dialog box.

7. Now place the annotative text:

a. Restart the **Text** command.

b. Use the **Style** option to change the text style to an annotative one.

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: s
Text style to use (or '?'): Annotative
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point)
```

c. Again place text with a height of 1/8":

```
Height of text <0.2000>: 0.125
Rotation angle of text <0>: (Press Enter)
Text: Hallway
Text: (Press Enter)
```


Notice that the text looks a reasonable size. ProgeCAD did the scale calculation for us and determined that the text should be 6" tall.

8. Any text you place now in Model tab will be scaled appropriately for the viewport and the scale factor. To see that the text appears at the correct size in the viewport, switch to **Layout1**. The text placed with the Standard style is essentially invisible, but the annotative text "Hallway" is correctly visible.
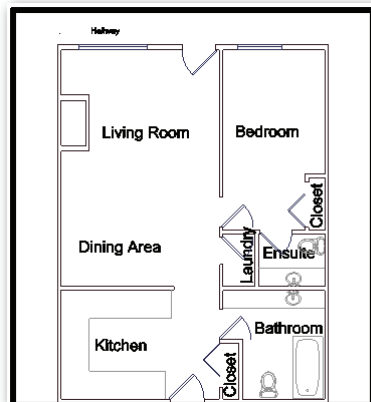
Editing the Scale List

The **ScaleListEdit** command lets you add and remove scale factors that you use in your drawings. This list of scale factors is used in three areas of progeCAD: annotative scales, viewport scales, and plot scales.

To add a scale factor, follow these steps:

    1.   Enter the **ScaleListEdit** command.

                Command: **scalelistedit**

       Notice the Edit Scale List dialog box.
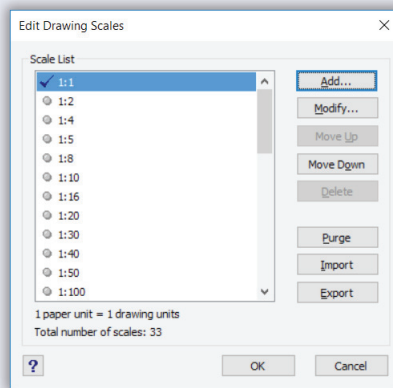
    2.   In the Edit Scale List dialog box, click **Add**.
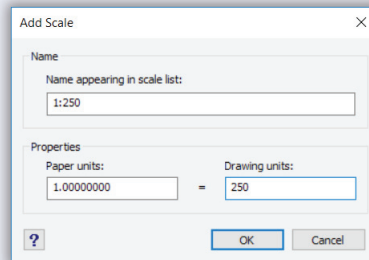
    3.   For the 'Name appearing in scale list' you can type anything, such as **1:250**.

    4.   Leave the Paper Units set to 1. For 'Drawing Units' enter **250**.

    5.   Click **OK**.

Notice that the new scale factor is added to the list. Use the **Move Up** and **Move Down** buttons to move commonly used scale factors to the top of the list.

9. Select the annotative text, and notice that the Properties bar reports the model text height of 6".
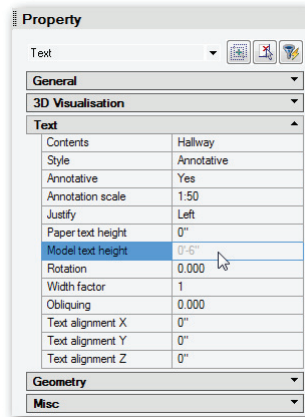
TIP   Should you need to plot the drawing at a second scale, then you must set up a second layout tab, set the second annotation scale, and then go back to model tab. In model tab, select the text and other annotations needing scaling, and then apply the new annotation scale factor.

Adding Dimensions

the floor plan drawing is almost complete. All that's missing are *dimensions*, which indicate lengths and distances. progeCAD supports a number of different kind of dimensions.

Linear dimensions measure the distance between two points. Most are horizontal or vertical, but some might be rotated (measured at an angle) or aligned (measured along an entity).

Radial dimensions measure the radius or diameter of curved entities, like arcs and circles.

Angular dimensions measure the angle between vertices.

Leaders are used to point text to entities.

Linear dimensions consist of these four parts:
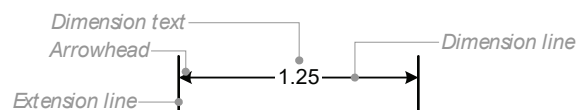
Dimension line — indicates the distance between the two points being measured

Dimension text — spells out the distance and its modifiers (if any)

Extension lines — link the dimension line with the points being measured

Arrowheads — traditionally point to the ends of the dimension line

In This Chapter

- **DimLinear** command
- **Dist** command
- **DimScale** command
- **List** command
- **DimStyle** command
- **Area** command

Hence, you typically pick three points for each linear dimension:

- One pick for the starting point of one extension line
- Another pick for the other extension line each extension line
- One pick that simultaneously locates the dimension line and the text

progeCAD figures out everything else: it draws the lines and arrowheads, and then measures the distance to generate the text. Later in this chapter you encounter the other dimension types. Like text, dimensions have styles that let you define the look and placement of every element of a dimension.

Linear Dimensions (DimLinear)

You place linear dimensions like this:

1. Open the floor plan drawing in progeCAD. (If you were unable to complete the drawing to this point, open the *4Text.dwg* file.)

2. Click the **Model** tab to return to model space. (If necessary, create some breathing space around the drawing with the **Zoom 0.8x** command.)

3. Change to the "Dimensions" layer using one of these methods:

- From the **Layer Control** droplist of the Entity Properties toolbar or Home tab of the ribbon, select "Dimensions."
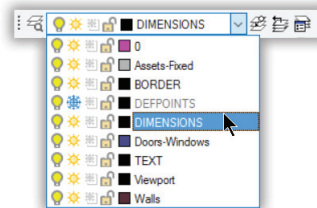
- Enter the **layer** command. When the Layer Properties Manager palette appears, select "Dimensions", and then click the **Current** ✔ button. Click **OK**.
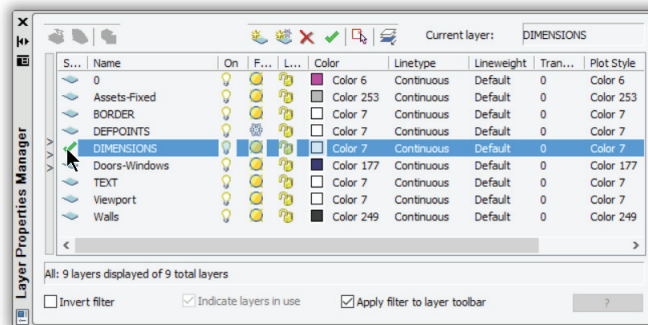
Dimensioning Commands

progeCAD provides the following commands for dimensioning entities:

DRAWING DIMENSIONS

DimAligned places dimensions aligned to entities.

DimAngular measure angles with dimensions.

DimArc dimensions the length of arcs.

DimBaseline continues from a previous linear, angular, or ordinate dimension; the first extension line is the start of the baseline dimension.

DimCenter places center markers at the center of circles, arcs, and polyline arcs.

DimContinue continues from a previous dimension; the final extension line of the last dimension is the start of the continued dimension.

DimDiameter measures the diameters of circles, arcs, or polyline arcs.

DimInspect adds inspection data to dimension text, such as percent completion.

DimJogged accommodates large-radius arcs and circles with a jogged dimension line.

DimLeader draws leaders.

DimLinear places horizontal and vertical dimensions.

DimOrdinate places X and Y dimensions, measuring them relative to the origin.

DimRadius dimensions the radii of circles, arcs, and polyline arcs.

DimRotated places linear dimensions that are rotated at a user-defined angle.

QLeader draws leaders, and provides a Settings dialog box.

QDim places a series of dimensions quickly, and is particularly useful for placing a series of baseline or continued dimensions, or for dimensioning a series of circles and arcs.

Tolerance creates G&DT boxes, geometric and dimensioning tolerances.

EDITING DIMENSIONS

In addition to using the commands listed below, dimensions can also be edited using grips.

DimEdit changes, rotates, and restore text, and obliques extension lines.

DimOverride overrides the style settings for selected dimensions.

DimTEdit changes the position and angle of dimension text.

SETTING AND CHANGING DIMENSION STYLES

DimStyle displays the Dimension Settings dialog box.

-DimStyle creates, changes, and saves dimension styles in the command bar.

- Use the **-Layer** command's **Set** option to change the working layer to "Dimensions," as follows:

  Command: **-layer**

  Layer: ? to list/New/Make/Set/Color/Ltype/LWeight/ON/OFF/Plot/Freeze/Thaw/LOck/Unlock: **s**

  Layer to set as current <0>: **dimensions**

  Layer: ? to list/New/Make/Set/Color/Ltype/LWeight/ON/OFF/Plot/Freeze/Thaw/LOck/Unlock: *(Press **Enter** to exit the command)*

4. To dimension the outside wall, start the **DimLinear** command:

   Command: **dimlinear**

5. progeCAD prompts you to take one of two directions: to select an entity, or else pick a location of the first extension line. For this tutorial, press **Enter** to choose the first option — select an entity to dimension:

   Specify origin of first extension line or ENTER to select entity: *(Press **Enter**)*
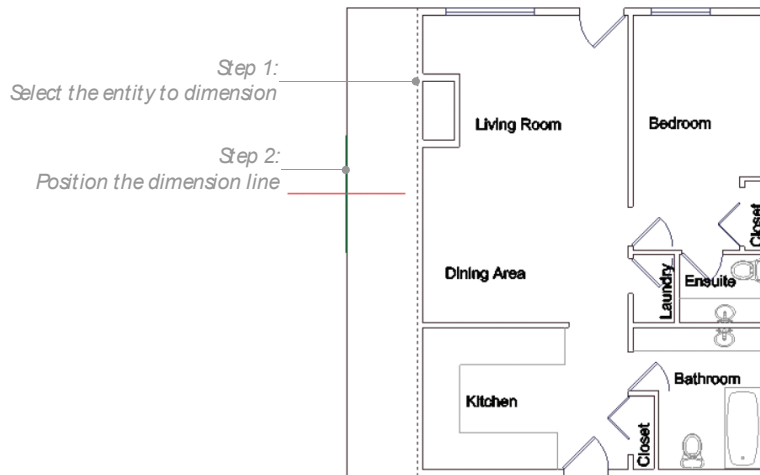
6. progeCAD asks which entity to dimension:

   Select entity to dimension: *(Pick the line at step 1)*

7. Finally, you need to tell progeCAD where to locate the dimension line:

   Specify dimension line location or [Angle/Mtext/Text/Horizontal/Vertical/Rotated]: *(Move cursor away from line, and then pick a point at step 2)*

TIP   The **DimLinear** command creates horizontal, vertical, or rotated dimensions — *based on the order of your pick points.* When you make the two pick points relatively horizontally, progeCAD draws a horizontal dimension; vertically, and vertical.

---

progeCAD draws the dimension, but look closely: can you see what's wrong?
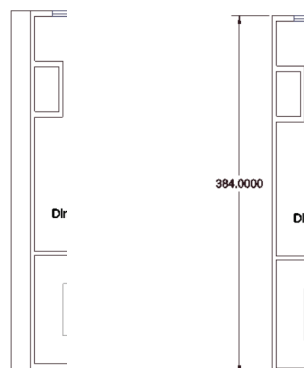
The arrowheads and dimension text appear to be missing, as shown by the figure at left. The problem is that they are too small to be seen, and should be bigger, as shown at right.

Dimension Scale (DimScale)

The solution to the problem of size is to scale the dimensions, just like text. The scaling is performed with the **DimScale** dimension variable.

1. Use the **U** command to remove the dimension you just drew.
2. Enter the **DimScale** command, and then use the same scale factor as for text in the previous chapter, 48.

   Command: **dimscale**
   Enter new value for DIMSCALE <1.0000>: **48**

3. Repeat the **DimLinear** command to re-dimension the line. This time the text and arrowheads are visible.

But the dimension text looks annoying with its decimal units and four decimal places. For an architectural drawing like this one, dimensions should be in architectural units — feet and fractional inches. The format problem is corrected with the **DimStyle** command.

Creates and edits dimension styles.

Command: **dimstyle**

Aliases: **d, ddim, setdim**

Menu: **Settings | Dimension Settings**

Toolbar: **Settings | Dimension Settings**

ALTERNATIVES

-DimStyle controls dimension styles at the command line.

ExpDimstyles displays the Explorer Dimension Styles window.

Dimension Styles (DimStyle)

The look of dimensions is set with the **DimStyle** command. Change the style so that dimension text is shown in feet and fractional inches.

1.   Start the **DimSytle** command.

Command: **dimstyle**

> Notice the Dimension Settings dialog box.
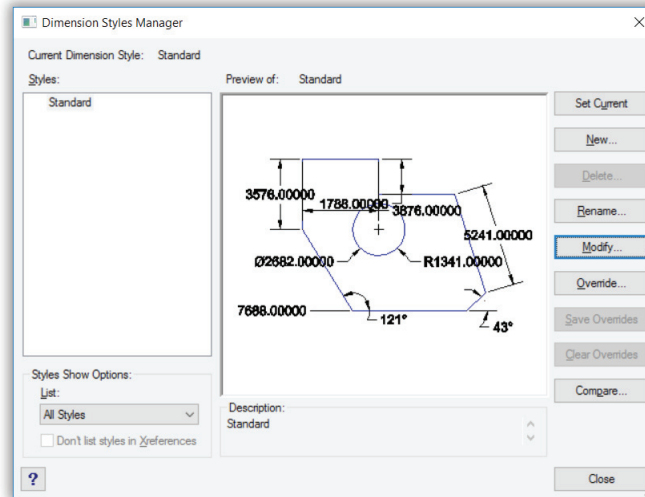
2.   The enormous preview window lets you see the effect of changing the dimension style. Take a quick tour of the complex dialog box by clicking on the tabs: Arrows, Format, Lines, and so on. This may well be the most imposing of all dialog boxes, but fortunately you need change only a few items.

(If necessary, select "Standard" from the **Dimension Style** droplist; ignore the warning.)

3.   Click the **Units** tab.
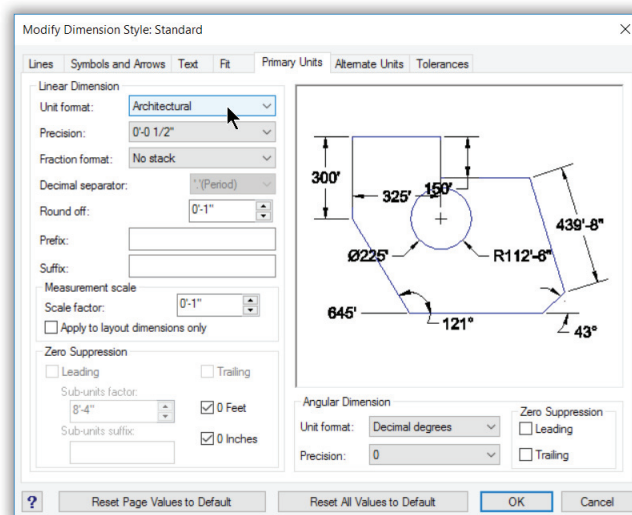
4. In the Linear Dimension section, click on the **Unit Format** drop list, and then select "Architectural."

5. Reduce **Precision** to **1/2"**. That means that fractional inches are dimensioned to the nearest 1/2-inch.

TIP   Use **Linear dimensioning scale factor** to switch between Imperial (inches) and metric measurement systems. For example, this drawing is in feet-and-inches. If you were to change this scale factor to **25.4**, all dimensions would change to millimeters.

---

6. Click **Save**. This forces the changes to be applied to the dimension style.

7. Click **OK**. Notice that dimension changes from 384.0000 to 32'. If it does not change, enter the **DimOverride** command to force the change.

Dimensioning Between Two Points

The other method of dimensioning is to pick the locations of the extension and dimension lines. To see how this works, dimension another part of the drawing.

1. Repeat the **DimLinear** command:
   > Command: **dimlinear**

2. Use INTersection entity snap to ensure the first extension line is placed at the a corner:
   > Specify origin of first extension line or ENTER to select entity: **int**
   > Snap to intersection of: *(Pick point at step#1)*
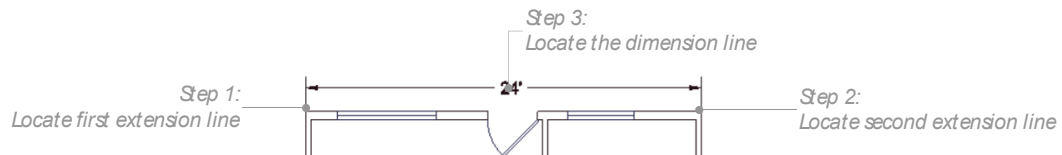
3. For the location of the second extension line, use INTersection esnap at the other corner:
   > Origin of second extension line: **int**
   > Snap to intersection of: *(Pick point at step #2)*

4. Now locate the dimension line:

> Specify dimension line location or [Angle/Mtext/Text/Horizontal/Vertical/Rotated]: *(Pick point at step #3)*

DIMCONTINUE COMMAND

Adds dimensions from the previous extension line.

Command: **dimcontinue**

Aliases: **dco, dimcont**

Menu: **Insert | Dimensions | Continue**

Toolbar: **Dimensions | Continue**

ALTERNATIVE

DimBaseline adds dimensions from the same starting point.

Continue Dimensions (DimContinue)

The right side of the floor plan has four rooms, and would use four dimensions. Here is where the **DimContinue** command comes in handy: it continues dimensioning from the previous one by placing the first extension line at the previous one, and then aligning the dimension line with the previous one.

Try it now to dimension the bedroom.

1. Start the DimContinue command.

Command: **dimcontinue**

2. Pick the location of the next extension line:

Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Pick a point)*

3. Continue until the end of the wall:

Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Pick another point)*

Continue: ENTER to select starting dimension/<Origin of next extension line>: *(Press **Esc** to exit the command)*

4. Dimension the lower wall. When you are done, the drawing should look something like the figure illustrated below.
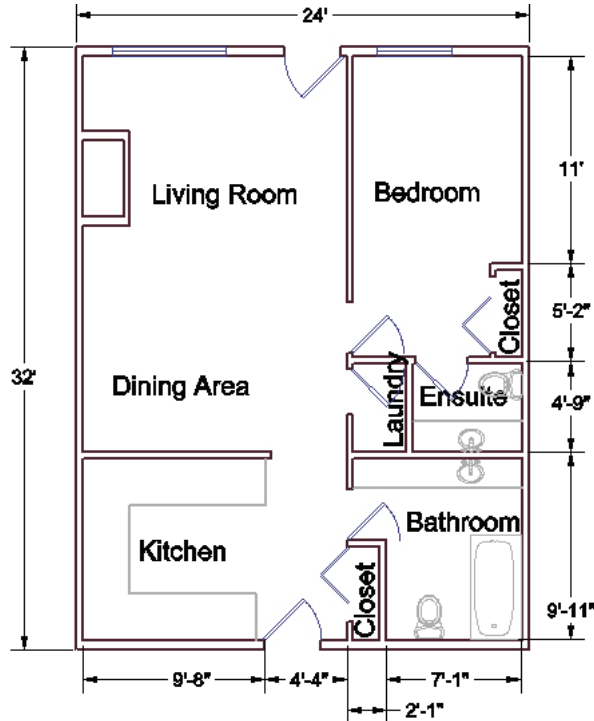
5. Save your work with the **QSave** command.

Other Methods of Measuring Drawings

Dimensioning commands are holdovers from the days when drawings were hand-drafted with pencils, straight edges, scale rulers, and erasers. (Image credit: Herbert Grabowski.) Back then, drawings were mostly drafted to scale. Dimensions helped others read the drawing without needing to reach for scale rulers — rulers in plural, because multiple rulers were needed for different scales and different disciplines, like architecture and engineering.
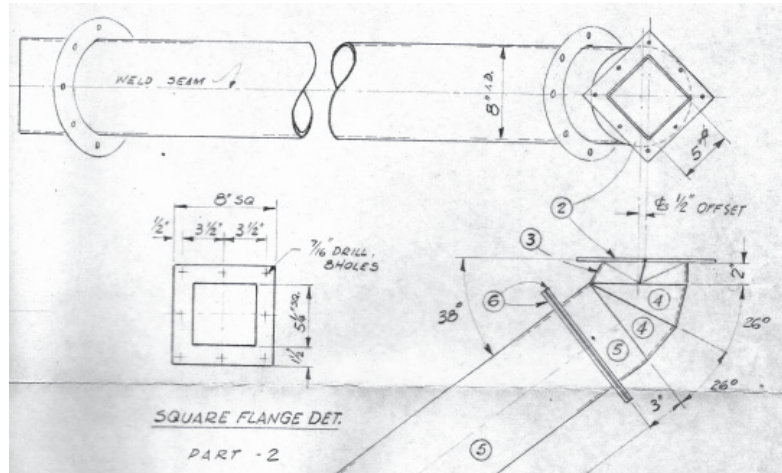
Strictly speaking, dimensions are not needed in drawings created by CAD software, because everything is drawn full-size. That 32'-long wall truly is thirty-two feet long. Scaling occurs only when it comes time to print the drawing. And so progeCAD includes commands for measuring distances, angles, and areas directly from drawings. Let's see how they work.

Measuring Distances and Angles (Dist)

The **Dist** command (short for "distance") measures distances and angles between two points. Use it to measure the length of the wall between the kitchen and dining area.

1.   Start the **Dist** command:
Command: **dist**

2.   As with dimensions, it's a good idea to use entity snaps to ensure the measurements are 100% accurate. (If necessary, turn on ENDpoint entity snap.)

progeCAD prompts you for two points:
Starting point for distance: *(Pick one end of the wall)*
End point: *(Pick other end of the wall)*

3.   Notice that progeCAD reports back that the wall is ten feet long:
Distance = 10'-0", Angle in XY Plane = 0, Angle from XY Plane = 0
Delta X = 10'-0", Delta Y = 0", Delta Z = 0"

If you cannot see all the text in the command bar, press **F2** to display the text window.

4. To see how angle measurements work, find the diagonal distance from one corner of the apartment to the opposite corner. Here is the result I got; your result may differ:

Command: **dist**

Starting point for distance: *(Pick one corner)*

End point: *(Pick other corner)*

Distance = 40'-1/2", **Angle in XY Plane = 233,** Angle from XY Plane = 0

Delta X = -24'-0", Delta Y = -32'-0", Delta Z = 0"

The meaning of the various distances and angles reported by the Dist command is illustrated by the following figure:
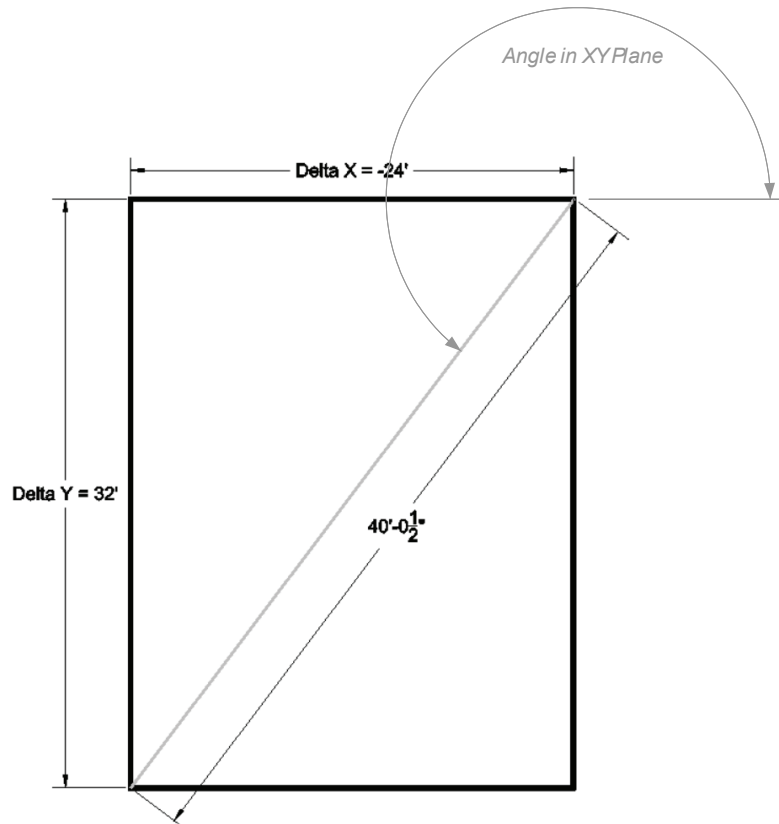
TIPS *Delta* means "difference" or the "net distance" between the two points.

Delta Z and **Angle from XY Plane** are used only for 3D models.

Reports everything pro-geCAD knows about entities.

Command: **list**

Aliases: **li, ls**

Menu: **Tools | Inquiry | List Entity Info**

Toolbar: **Inquiry | List Entity Info**

ALTERNATIVES

Area reports the area and perimeter of closed entities.

Dist measures the distance between two points.

Measuring Entities (List)

The **List** command reports information about selected entities. It returns everything progeCAD knows about the entities; much of that information isn't terribly useful to everyday drafting; still, this command is handy when you can't figure out what's happening with some entity or another.

In the context of this chapter, **List** returns length, perimeter, circumference, and area data — the data varies, depending on the entity. Try out the command on several entities.

1.   Start the **List** command:
Command: **list**

2.   Select any line in the drawing.
SCroll/SOrt/Tracking/<Select entities to list>: *(Pick a line in the drawing)*

3.   progeCAD displays its report in the text window, because the listing is longish:

```
---------- Line --------------------------------------------------
    Handle: 55
Current space:          Model
      Layer: Walls
      Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:  30
From point:  X=   2'-6"  Y=   27'-0"  Z=   0"
  To point: X=   2'-6"  Y=   23'-0"  Z=   0"
   Length: 4'-0"
Angle in UCS XY plane:          270
Delta values: X=   0"  Y=   -4'-0"  Z=   0"
```

In the listing above, I highlighted the length of four feet.

4.   Now select a block with the List command. Here is what the listing looks like:

```
---------- Insert of Block --------------------------------------
    Handle: 212
Current space:          Model
      Layer: Assets-Fixed
      Color: 256 (BYLAYER)
 Linetype: ByLayer
Lineweight:  30
Block name: Tub-Topview
Insertion point:        X=   23'-7"  Y=   3'-4"  Z=   0"
X scale factor:         1
Y scale factor:         1
Z scale factor:         1
Rotation angle:         90
```

5. And then select a piece of text with the List command:

```
---------- Text -------------------------------------------------
      Handle: 6C1
Current space:        Model
     Layer: BORDER
     Color: 256 (BYLAYER)
  Linetype: ByLayer
Lineweight:  0
     Color: 256 (BYLAYER)
  Linetype: ByLayer
Lineweight:  0
Text style: Standard
Insertion point:       X=  1'-11" Y=  13'-7" Z=  0"
Text height:  1'-0"
Text value:   Dining Area
Rotation angle:        0
Generation:  Normal
Vertical alignment:    Baseline
Horizontal alignment: Left
```

Measuring Areas (Area)

The **Area** command finds the areas of entities and spaces. Here's how to find the area of any room.

1.   Start the **Area** command:

Command: **area**

2.   With the assistance of INTersection entity snap, pick each corner of the Ensuite bathroom.

Entity/Add/Subtract/<First point>: *(Pick one corner)*

<Next point>: *(Pick another corner)*

<Next point>: *(Pick the third corner)*

<Next point>: *(Pick the last corner)*

1. First point
4. Fourth point
2. Second pick point
3. Third point
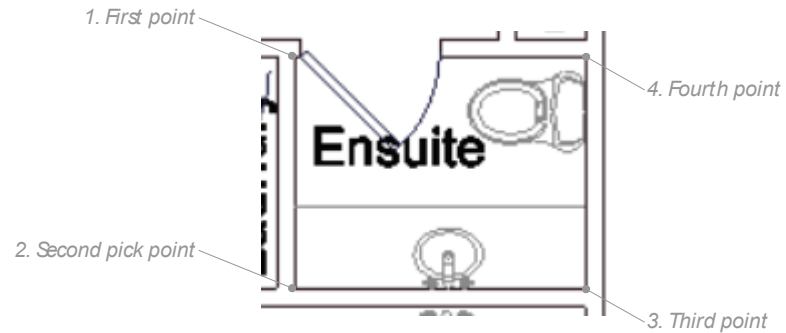
3. Press **Enter** to end the measurement process. progeCAD reports the result:

<Next point>: *(Press Enter to exit the command)*

Area = 3920" sq. (**27' sq.**), Perimeter = 21'-0"

The bathroom has an area of twenty-seven square feet. This tells the subcontractor the amount of floor tile to order. And the perimeter of twenty-one linear feet is the length of baseboard material that will be needed.

TIP   The **Area** command's **Add** and **Subtract** options allow you to add other areas and remove "holes" to find a grand total.

The command's **Entity** option reports the area and perimeter of closed entities: circles, ellipses, polylines, polygons, traces, and planes. In the case of polylines, Area "closes" the start and end points to find the area.

The Area command also with open entities, but only the length of lines, arcs, and splines is reported.

Interacting with Satellite Images

Y ou can place your drawings on top of maps and aerial views in progeCAD, and publish drawings to Google Earth. To do any of this, you first need know the position of your drawing on the earth's surface so that it is located correctly in terms of longitude and latitude.

progeCAD read map and satellite photo files in many format, including the following:

EDW — proprietary compressed image format developed for aerial imagery by Earth Resource Mapping

MrSid — proprietary format patented by LizardTech that encodes geo-referenced raster graphics, (short for multi resolution seamless image database)

Earth — image tiles provided by Google Earth

Most of these images are raster files that are placed in the drawing using the **ImageAttach** command; images from Earth are placed using the **ImportGeImage** command.

In This Chapter

- Setting up Google Earth
- Exporting to Google Earth
- Adjusting the image and drawing
- Placing MrSid satellite images

Placing Satellite Images from Google Earth

The method I consider the easiest is to import an area from Google Earth into progeCAD. The area imported is the current Google Earth view; it is imported into progeCAD as a JPEG raster image. You use the raster image as a background to a 2D drawing or 3D model.

Let's dive right in and place the floor plan drawing on a satellite image from Google Earth. Here are the steps involved:

1.   Start Google Earth. (If necessary, download and install Google Earth from https://www.google.com/earth/download/ge/agree.html on your Windows or Mac computer.)

2.   Maximize the Google Earth window. Do not minimize or close the Google Earth application at any time during this tutorial!

3.   Zoom to the location on the earth's surface on which you wish to place your drawing. Zoom in to the satellite view to capture enough detail of the location area.

4.   Be sure you are looking down perpendicularly to the earth's surface (the view has no rotation). At this point, you might be seeing something like the figure shown below.
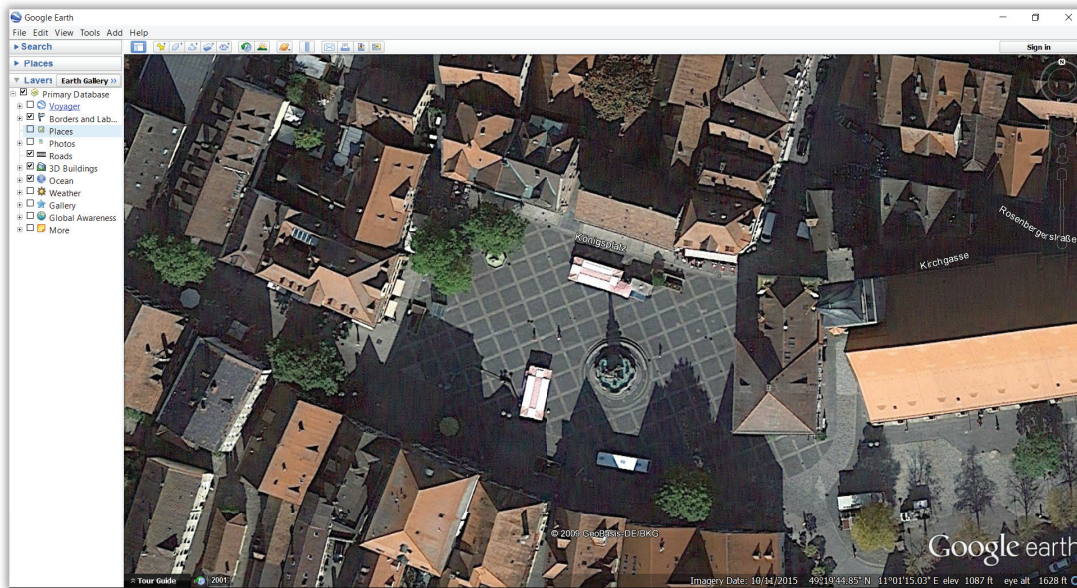
5.   Switch to progeCAD, and then open your drawing. If it is a 3D model, then ensure it is in plan view.

6.   Enter the **ImportGeImage** command.

7. Notice the dialog box: choose metric or Imperial drawing units to scale the image correctly to your drawing.
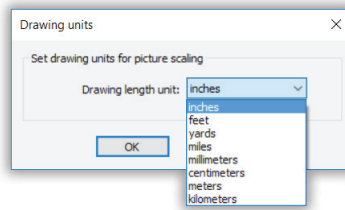
8. Click **OK**. After a few seconds, the current Google Earth view is imported and placed at the origin (0,0) of the drawing. Notice that the image is black and white; this is a limitation imposed by Google for using these images free.
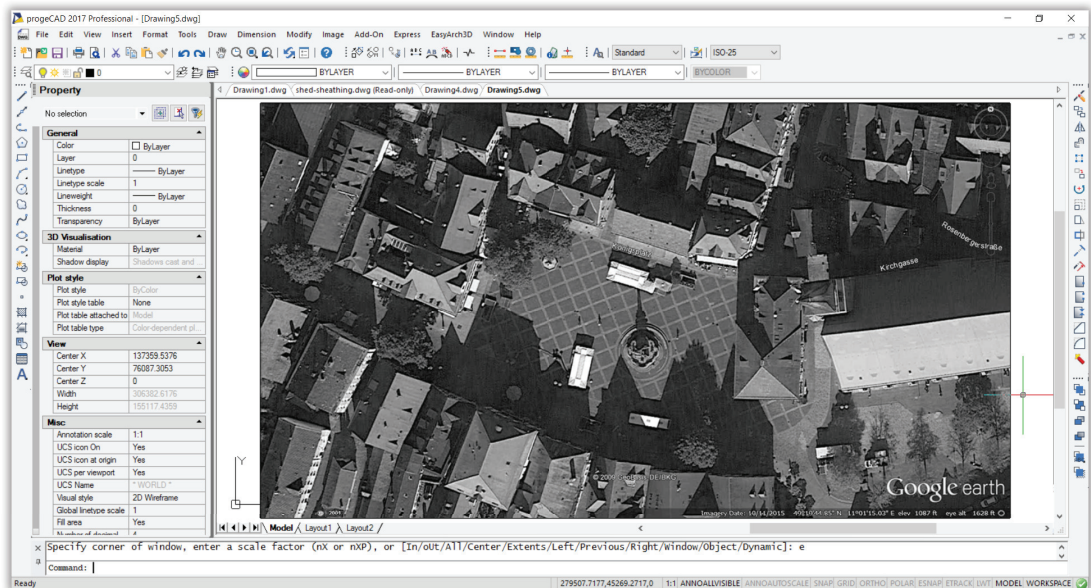
If you cannot see anything, use **Zoom Extents** to include all objects of the drawing.

Working With Google Earth Images

When working with images from Google Earth, the following suggestions may help:

- Once the image is placed, you can use progeCAD's **Move** and **Rotate** commands to refine the drawings's position relative to the image. Do not adjust the position of the Google Earth's image!
- The image displayed in the drawing is a JPG file and it is stored in the same directory as the DWG file.
- If necessary, use the **DrawOrder** command to visually place the image underneath the drawing, like this: enter the **DrawOrder** command, select the image, and then press Enter. Specify the new drawing order, and then press Enter.
- Images from Google Earth are only 2D; it does not display in 3D, so no contours or 3D buildings are available.

Placing Mapping Imagery from Other Sources

progeCAD also loads mapping and satellite images from many other sources. For the complete list, see the "All Supported Files" list in the **ImageAttach** dialog box:

All supported files (*.*)
BMP (*.BMP;*.DIB;*.RLE)
EMF (*.EMF)
GIF (*.GIF)
ICO (*.ICO)
JPEG (*.JPG;*.JPEG;*.JPE;*.JFIF)
PNG (*.PNG)
TIFF (*.TIF;*.TIFF)
WMF (*.WMF)
MrSID (*.sid)
ADRG/ARC Digitized Raster Graphics (*.gen;*.thf)
Arc/Info Binary Grid (*.adf)
ArcInfo ASCII Grid (*.asc)
Aster/Modis/Landsat7/EODIS/Spot Vegetation (*.hdf;*.met;*.l1r;*.l1g)
BSB Nautical Chart Format (*.kap)
ECW (*.ecw)
ER Mapper Algorithm (*.alg)
ER Mapper Raster Dataset (*.ers)
ERDAS Imagine (*.img)
ESRI BIL and GeoSPOT (*.bil)
Envisat Image Product (*.n1)
FITS (*.fits)
InfoSAR NetCDF Images (*.nc)
JPEG (*.jpg)
JPEG2000 (*.jp2;*.j2k;*.j2c;*.jpc;*.jpx;*.jpf)
Japanese DEM (*.mem)
Landsat 7 FastL7A (*.fst)
Military Elevation Data (*.dt0;*.dt1)
National Imagery Transmission Format (*.ntf)
Netpbm (*.ppm;*.pgm)
RESTEC/NASDA CEOS (*.dat)
Sandia National Laboratories GSAT File Format (*.gff)
TIFF (*.tif;*.tiff)
USGS ASCII DEM (*.dem)
USGS Digital Ortho Quad (*.doq)
USGS SDTS DEM (*.ddf)
ACE2 (*.ACE2)
ARC Digitized Raster Graphics (*.gen)
ASCII Gridded XYZ (*.xyz)
Arc/Info ASCII Grid (*.asc)
Arc/Info Export E00 GRID (*.e00)
DRDC COASP SAR Processor Raster (*.hdr)
ECRG TOC format (*.xml)
ENVI .hdr Labelled (*.*)
ERDAS Compressed Wavelets (SDK 5.1) (*.ecw)
ERDAS JPEG2000 (SDK 5.1) (*.jp2)
EUMETSAT Archive native (.nat) (*.nat)
Envisat Image Format (*.n1)
Erdas Imagine Images (.img) (*.img)
FARSITE v.4 Landscape File (.lcp) (*.lcp)
FIT Image (*.*)
GMT NetCDF Grid Format (*.nc)
GRIdded Binary (.grb) (*.grb)
GeoSoft Grid Exchange Format (*.gxf)
GeoTIFF (*.tif)
Geospatial PDF (*.pdf)
Golden Software 7 Binary Grid (.grd) (*.grd)
Golden Software ASCII Grid (.grd) (*.grd)
Golden Software Binary Grid (.grd) (*.grd)
Graphics Interchange Format (.gif) (*.gif)
Graphics Interchange Format (.gif) (*.gif)
Ground-based SAR Applications Testbed File Format (.gff) (*.gff)
HF2/HFZ heightfield raster (*.hf2)
Hierarchical Data Format Release 4 (*.hdf)
Hierarchical Data Format Release 5 (*.hdf5)
ILWIS Raster Map (*.mpr/mpl)
IRIS data (.PPI`.CAPPi etc) (*.ppi)
Idrisi Raster A.1 (*.rst)
JPEG JFIF (*.jpg)
JPEG-2000 driver based on OpenJPEG library (*.jp2)
Japanese DEM (.mem) (*.mem)
KOLOR Raw (*.kro)
Leveller heightfield (*.ter)
MBTiles (*.mbtiles)
MS Windows Device Independent Bitmap (*.bmp)
Magellan topo (.blx) (*.blx)
MrSID Generation 4 / Lidar (.sid) (*.view)
Multi-resolution Seamless Image Database (MrSID) (*.sid)
NOAA NGS Geoid Height Grids (*.bin)
NOAA Vertical Datum .GTX (*.gtx)
NTv2 Datum Grid Shift (*.gsb)
Network Common Data Format (*.nc)
Northwood Classified Grid Format .grc/.tab (*.grc)
Northwood Numeric Grid Format .grd/.tab (*.grd)
PCIDSK Database File (*.pix)
PCRaster Raster File (*.map)
Portable Network Graphics (*.png)
Portable Pixmap Format (netpbm) (*.pnm)
R Object Data Store (*.rda)
Raster Matrix Format (*.rsw)
Raster Product Format TOC format (*.toc)
Rasterlite (*.sqlite)
SAGA GIS Binary Grid (.sdat) (*.sdat)
SDTS Raster (*.ddf)
SGI Image File Format 1.0 (*.rgb)
SRTMHGT File Format (*.hgt)
Snow Data Assimilation System (*.hdr)
Standard Raster Product (ASRP/USRP) (*.img)
Swedish Grid RIK (.rik) (*.rik)
Terragen heightfield (*.ter)
USGS Optional ASCII DEM (and CDED) (*.dem)
VTP .bt (Binary Terrain) 1.3 Format (*.bt)
Vexcel MFF Raster (*.hdr)
Virtual Raster (*.vrt)
X11 PixMap Format (*.xpm)
ZMap Plus Grid (*.dat)

They are, however, not as handy for non-experts as Google Earth. First, you have to locate the sources for the images; second, you might quite possibly have to pay for them. A Web page with free samples from a large number of satellite services can be found at http://cms.mapmart. com/samples.aspx, as shown below.
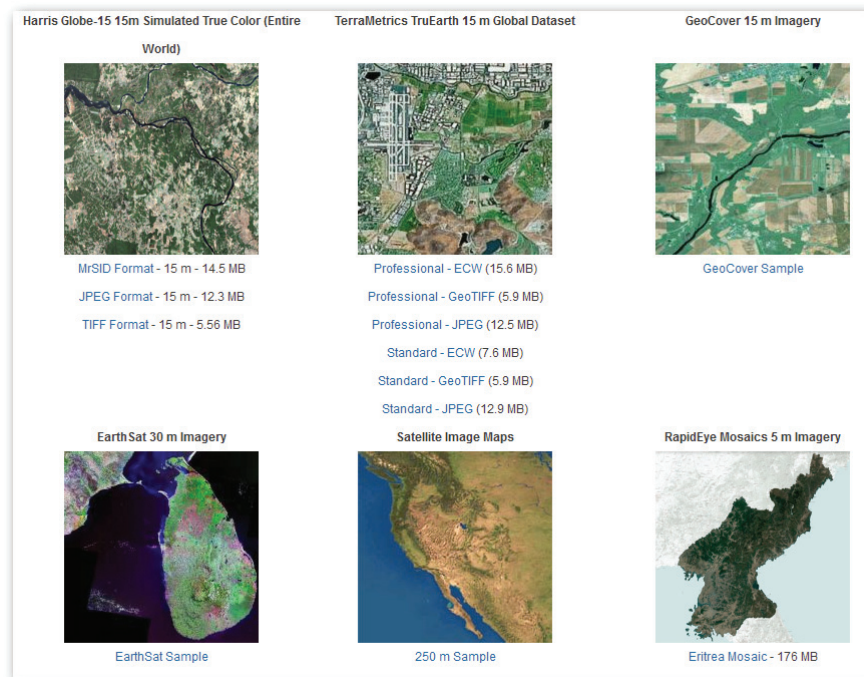
TIP   A single satellite image file or 3D terrain file can be very large, such as a file hundreds of megabytes in size. This can slow down operations of a CAD system.

In the following tutorial, you place a satellite image saved in MrSid format:

1.   Obtain a MrSid file, such as from the source listed above. It probably is a ZIP file holding two files:

      .sid — the satellite image

      .sdw — the image positioning file

2.   Enter the **ImageAttach** command.

3.   Notice the Select Image File dialog box. Navigate to the folder holding the *.sid* MrSid file, select it, and then click **Open**.
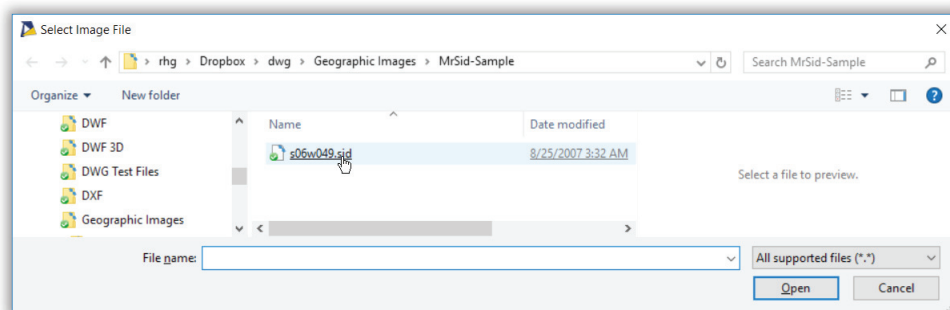
4. Notice the Attach Image dialog box, and then choose the Positioning File, as follows:

   a.   Turn on the **Use Positioning File** option.
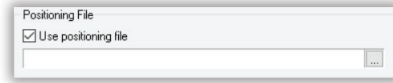
   b.   Click the [...] **Browse** button. Notice the Open dialog box.
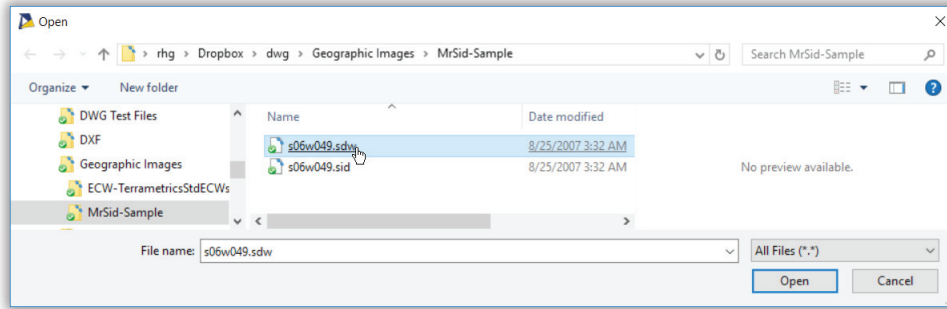
   c.   Click the "File type" droplist, and then choose **All Files**.

   d.   In the same folder from which you opened the *.sid* file, choose the matching *.sdw* file. It holds the geographic coordinates of the satellite image.

   e.   Click **Open**. Notice that the Insertion Point coordinates are filled in for you.
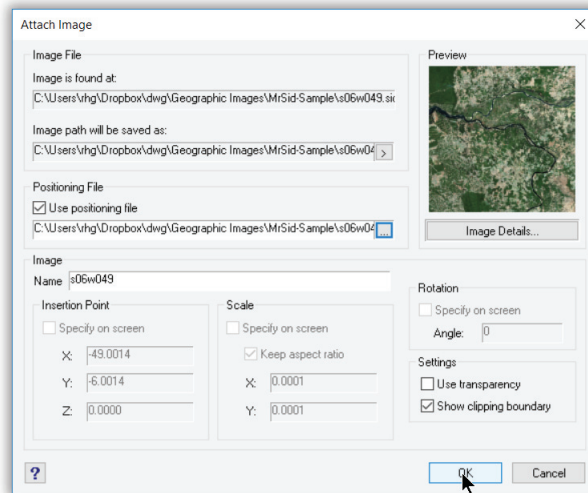
5. Click **OK** to place the image in your drawing. You may need to Zoom Extents to see the full image.
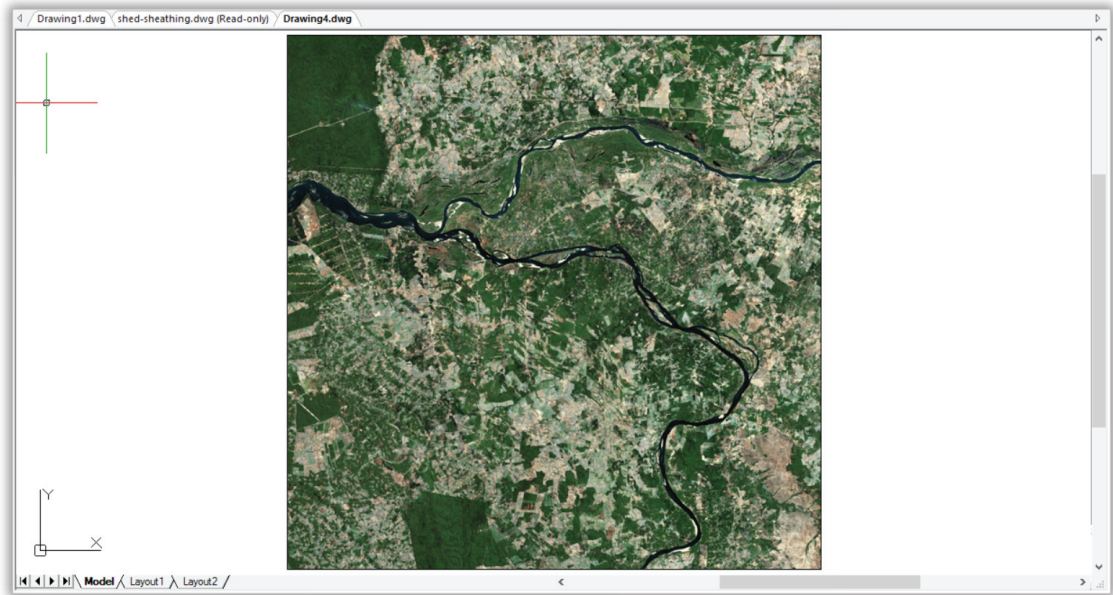
TIP   To place a 3D mesh of landscapes in drawings, look for DXF files, such as the one imported into progeCAD below. It is 243MB in size.

Publishing Drawings to Google Earth

To publish drawings to Google Earth, follow these steps:

1.   Enter the **PublishKML** command. Notice the Export to Google Earth dialog box.
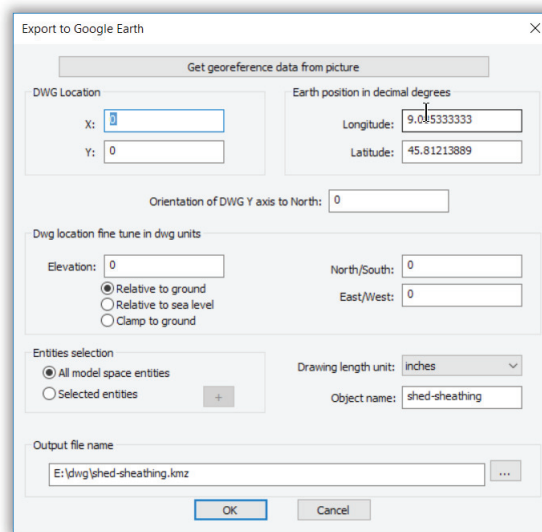
2.   In the dialog box, click the **Get Georeference Data from Picture**.

3.   Select the Google Earth image that you previously inserted. This picks up the coordinates.

4. If the position and rotation of a model relative to the image are correct, there's no need to modify the orientation of the Y axis to North.

   The output file name is taken automatically from the DWG file name and directory.

5. Click **OK**. progeCAD generates a *.kmz* file.

TIP   The KMZ file generated by export has the same name and it is stored in the same folder as the DWG file. (KMZ is short for "keyhole model compressed." Keyhole is the name of the company from which Google purchased Earth.)

Colors of published entities in Google Earth match progeCAD's object colors.

---

6. Go to the folder that holds the *.kmz* file, and then click on the file. It will be opened in Google Earth, and the software will zoom into the place on the planet at which the drawing is situated. If necessary to see the drawing better, you may need to zoom in or out using Google Earth's zoom slider.
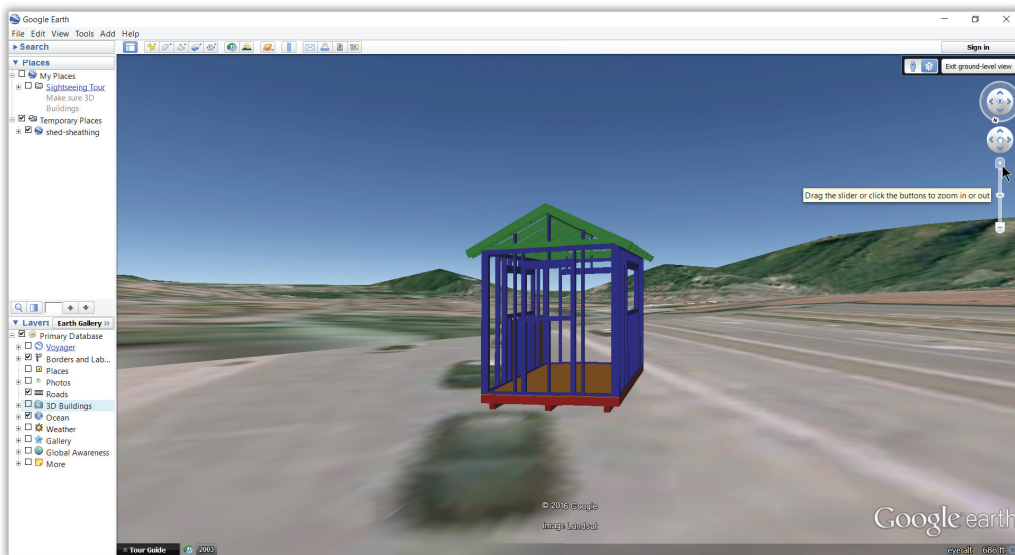
If the drawing you exported is a 3D model, then it will appear in 3D on Google Earth.

Keyboard Shortcuts, Aliases & Buttons

this appendix lists the command names included with progeCAD. It also has the default keyboard shortcuts and command aliases in alphabetical order. And it provides a list describing actions of mouse buttons.

In This Appendix

• Keyboard Shortcuts   • Command Aliases   • Mouse Buttons

Keyboard Shortcuts

Keyboard shortcuts are abbreviations for commands. They consist of function keys, or holding down the Ctrl and Shift keys in conjunction with other keys. Keyboard shortcuts are created and changed with the **Customize** command.

Shortcut	Meaning	Command Executed
Function Keys		
F1⊞	Display on-line help	HELP
F2	Display Prompt History window	PMTHIST
F3	Toggle object snap mode	ESNAP T
Alt+F3	Close current drawing	Close
Alt+Shift+F3	Close all open drawings	CloseAll
F4	Toggle tablet mode	TABLET T
F5	Switch to next isoplane	ISOPLANE
F6	Toggle dynamic UCS	DYNUCS T
F7	Toggle display of the grid	GRID T
F8	Toggle ortho mode	ORTHOGONAL T
Alt+F8⊞	Run VBA macro	VBARUN
F9	Toggle snap mode	SNAP T
F10	Toggle polar tracking	POLARTRACK T
Alt+F10	Toggle status bar	STATBAR T
F11	Toggle entity tracking	ENTTRACK T
Alt+F11⊞	Start VBA Manager	VBA
F12	Toggle dynamic input	DYNINPUT T
Control Keys		
Ctrl+A⊞	Select all entities in drawing	SELGRIPS ALL
Ctrl+C⊞	Copy selected entities to Clipboard	COPYCLIP
Ctrl+Shift+C	Copies entities with a base point	COPYBASE
Ctrl+E	Switch to next isoplane	ISOPLANE
Ctrl+I	Toggle coordinates	COORDINATE T
Ctrl+N⊞	Start a new drawing with wizard	PNEW
Ctrl+O⊞	Open a drawing file	OPEN
Ctrl+P⊞	Print the drawing	PRINT
Ctrl+Q	Exit the program	EXIT
Ctrl+R	Switch to next viewport	^V
Ctrl+S⊞	Save the drawing	QSAVE
Ctrl+Shift+S	Prompt for file name and save	SAVEAS
Ctrl+T	Toggle tablet mode	TABLET T
Ctrl+V⊞	Paste from Clipboard into drawing	PASTECLIP
Ctrl+Shift+V	Pastes Clipboard contents as a block	PASTEBLOCK
Ctrl+X⊞	Cut selected entities to Clipboard	CUTCLIP
Ctrl+Y⊞	Redo the last undo	REDO
Ctrl+Z⊞	Undo the last command	U
Ctrl+1	Toggle the Properties palette	PROPERTIES
Ctrl+2	Toggle the Layer palette	LAYER
Ctrl+9	Toggle the Command Bar	CMDBAR T

Other Keys

Esc	Cancels current command	...
Del⌨	Deletes selected entities	DELETE
Left Arrow	Pans left	PAN L
Right Arrow	Pans right	PAN R
Down Arrow	Pans down	PAN D
Up Arrow	Pans up	PAN U
Shift+Left Arrow	Pans left by a page full	PAN PGL
Shift+Right Arrow	Pans right by a page full	PAN PGR
Shift+Down Arrow	Pans down by a page full	PAN PGD
Shift+Up Arrow	Pans up by a page full	PAN PGU
Page Down	Pans down by a page full	PAN PGD
Page Up	Pans up by a page full	PAN PGU

⌨ Shortcut key standard in Windows.

Command Aliases

Aliases are one-, two-, and three-letter abbreviations for commands. They let keyboardists enter commands faster; it is far quicker to type "l" than to type "line".

Aliases can be used to map the names of commands from other CAD packages, such as AutoCAD, to progeCAD commands. For example, AutoCAD's "pedit" is mapped to progeCAD's EditPline command. Aliases cannot be used for command options, or for accessing options within dialog boxes. Aliases are created and changed with the **Customize** command.

Alias	Command	Alias	Command
?	HELP		
		BM	BLIPMODE
		BO	BOUNDARY
A		-BO	-BOUNDARY
A	ARC	BR	BREAK
AA	AREA		
AD	ATTDISP	**C**	
AL	ALIGN	C	CIRCLE
AP	APERTURE	-CH	CHANGE
-ARRAY	-ARRAY	CHA	CHAMFER
AT	ATTDEF	CL	COPYLINK
-AT	-ATTDEF	CLIP	XCLIP
ATE	ATTEDIT	CP	COPY
ATT	ATTDEF		
-ATT	-ATTDEF	**D**	
AUTOCOMPLETE	-INPUTSEARCHOPTIONS	D	-DIMSTYLE
AX	-ATTEXT	DAL	DIMALIGNED
		DAN	DIMANGULAR
B		DBA	DIMBASELINE
B	-BLOCK	DCE	DIMCENTER
BA	BASE	DCO	DIMCONTINUE
BHATCH	HATCH	DDATTEXT	ATTEXT
-BHATCH	-HATCH	DDCHPROP	PROPERTIES
-BLOCK	-BLOCK	DDFILTER	SELECT

Alias	Command	Alias	Command
DDI	DIMDIAMETER	GR	DDGRIPS
DDIM	-DIMSTYLE	**H**	
DDNEW	NEWWIZ	H	-HATCH
DDRENAME	EXPLAYERS	HE	-HATCHEDIT
DED	DIMEDIT	HI	HIDE
DELETE	ERASE		
DI	DIST	**I**	
DIM	DIMENSION	I	INSERT
DIMALI	DIMALIGNED	IAD	OPENIMAGE
DIMANG	DIMANGULAR	IAT	IMAGEATTACH
DIMBASE	DIMBASELINE	ICL	IMAGECLIP
DIMCONT	DIMCONTINUE	ID	IDPOINT
DIMDIA	DIMDIAMETER	IMP	IMPORT
DIMED	DIMEDIT	IN	INTERSECT
DIMLEADER	DIMLEADER	INF	INTERFERE
DIMLIN	DIMLINEAR	INFLINE	XLINE
DIMORD	DIMORDINATE	IO	INSERTOBJ
DIMOVER	DIMOVERRIDE	IS	ISOPLANE
DIMRAD	DIMRADIUS		
DIMTED	DIMTEDIT	**L**	
DIV	DIVIDE	L	LINE
DL	MLINE	LA	EXPLAYERS
DLI	DIMLINEAR	-LA	-LAYER
DLINE	MLINE	LE	LEADER
DO	DONUT	LEN	LENGTHEN
DOR	DIMORDINATE	LI	LIST
DOV	DIMOVERRIDE	LS	LIST
DRA	DIMRADIUS	LT	EXPLTYPES
DS	DDRMODES	-LT	-LINETYPE
DST	-DIMSTYLE	LTS	LTSCALE
DT	TEXT		
DV	DVIEW	**M**	
DXI	DXFIN	M	MOVE
DXO	_EATTEXT	MA	MATCHPROP
		MI	MIRROR
		MO	PROPERTIES
		MS	MSPACE
E		MSLIDE	MSLIDE
E	ERASE	MT	MTEXT
ED	DDEDIT	MV	MVIEW
EDITLEN	LENGTHEN		
EL	ELLIPSE	**N**	
EX	EXTEND	N	NEW
EXP	EXPORT		
EXPLORER	EXPLAYERS	**O**	
EXPSTYLE	EXPFONTS	O	OFFSET
EXPSTYLES	EXPFONTS	OO	UNDELETE
EXT	EXTRUDE	OP	CONFIG
		-OS	ESNAP
F		OS	SETESNAP
F	FILLET		
FI	SELECT	**P**	
		-P	-PAN
G		P	RTPAN
G	-GROUP		

Alias	Command	Alias	Command
PA	PASTESPEC	**T**	
PE	EDITPLINE	T	MTEXT
PL	POLYLINE	-T	TEXT
PO	POINT	TA	TABLET
POL	POLYGON	TI	_TILEMODE
PR	PROPERTIES	TO	TBCONFIG
PRE	PPREVIEW	TOL	TOLERANCE
PROPERTIES	PROPERTIES	TOR	TORUS
PS	PSPACE	TR	TRIM
PU	-PURGE	TX	TEXT
Q		**U**	
QT	QTEXT	UC	EXPUCS
		UCP	SETUCS
R		UN	DDUNITS
R	REDRAW	-UN	-UNITS
RA	REGENALL	UNI	UNION
RCONFIG	CONFIG	URL	BROWSER
RE	REGEN		
REA	REGENALL	**V**	
REC	RECTANGLE	V	VIEW
RECT	RECTANGLE	-V	-VIEW
RECTANG	RECTANGLE	VL	VPLAYER
REG	REGION	VP	VIEWCTL
REN	EXPLAYERS	-VP	VIEWPOINT
REV	REVOLVE	VS	VSCURRENT
RI	REINIT	VSTAIDE	VSTA
RM	DDRMODES	VW	VPORTS
RO	ROTATE		
RPREF	SETRENDER	**W**	
RR	RENDER	WB	-WBLOCK
		WE	WEDGE
S		WI	WMFIN
S	STRETCH	WO	WMFOUT
SA	SAVE		
SC	SCALE	**X**	
SCR	SCRIPT	X	EXPLODE
SE	DDSELECT	XA	-XREF
SEC	SECTION	XB	XBIND
SEL	SELECTSIMILAR	-XB	XBIND
SELGRIP	SELGRIPS	XL	XLINE
SET	SETVAR	XR	-XREF
SETDIM	-DIMSTYLE	-XR	-XREF
SETLANG	CODEPAGE	XRM	-XREF
SHA	SHADE		
SL	SLICE	**Z**	
SN	SNAP	Z	ZOOM
SO	SOLID	**#**	
SP	SPELL		
SPE	SPLINEDIT	3A	3DARRAY
SPL	SPLINE	3DO	3DORBIT
ST	FONT	3F	FACE
SU	SUBTRACT	3P	3DPOLY

Mouse Buttons

When mouse buttons are clicked, they execute commands in progeCAD. Mouse buttons cannot be customized.

Button	Action
Objects Not Selected	
Left	Selects entities, menu items, toolbar buttons, and so on
Shift+Left	Displays shortcut menu with entity snaps
Right	Ends command; repeats last command
Entities Selected	
Left (single click)	Deselects entities
Left (double-click)	Displays entity properties dialog box
Shift+Left	Displays shortcut menu with entity snaps
Right	Displays shortcut menu with editing commands
Wheel (scroll bars not displayed)	
Roll forward	Zooms in 1.25x
Roll back	Zooms out 0.8x
Wheel (scroll bars displayed)	
Roll forward	Pans up
Roll back	Pans down
Ctrl+Shift+Left	Pans left or right
Shift+Left	Zooms in or out
Other Commands	
Left	Starts and ends sketch mode (Sketch command)
Left	Enables real-time rotation (RtRot command)